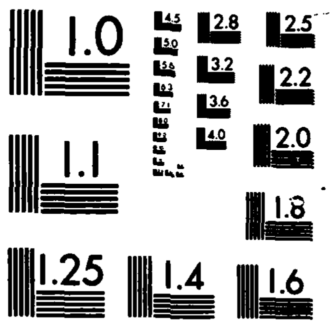
AD-A163 258 CADS - A COMPUTER AIDED DESIGN SYSTEM VOLUME 2 USER'S GUIDE(U) ROCKWELL INTERNATIONAL LOS ANGELES CA NORTH AMERICAN AIRCRAFT.. M C LESS ET AL. AUG 85 1/3

UNCLASSIFIED AFWAL-TR-85-3066-VOL-2 F33615-81-C-3229 F/G 9/2 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

12

AFWAL-TR-85-3066
Volume II

CADS - A COMPUTER AIDED DESIGN SYSTEM
Volume II - User's Guide

Michael C. Less
Susan Manual

Rockwell International
North American Aircraft Operations (NAAO)
El Segundo, California 90009

August 1985

Final Report for Period December 1981 - May 1985

FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture use, or sell any patented invention that may in any way be related thereto.

This report has been reviewed by the Office of Public Affairs (ASD/PA) and is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

This technical report has been reviewed and is approved for publication.

Victoria A. Tischler
VICTORIA A. TISCHLER
Project Engineer
Design & Analysis Methods Group

Fred A. Picchioni
FREDERICK A. PICCHIONI, Lt Col, USAF
Chief, Analysis & Optimization Branch

FOR THE COMMANDER

R J Hegstrom
ROGER J. HEGSTROM, Col, USAF
Chief, Structures & Dynamics Div.

"If your address has changed, if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization please notify AFWAL/FIBRA, W-PAFB, OH 45433 to help us maintain a current mailing list".

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

AD-A163 258

REPORT DOCUMENTATION PAGE

1a. REPORT SECURITY CLASSIFICATION: UNCLASSIFIED	1b. RESTRICTIVE MARKINGS: N/A
2a. SECURITY CLASSIFICATION AUTHORITY: N/A	3. DISTRIBUTION/AVAILABILITY OF REPORT: Approved for Public Release; Distribution Unlimited.
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE: N/A	
4. PERFORMING ORGANIZATION REPORT NUMBER(S)	5. MONITORING ORGANIZATION REPORT NUMBER(S): AFWAL-TR-85-3066
6a. NAME OF PERFORMING ORGANIZATION: Rockwell International; 6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION: Air Force Wright Aeronautical Laboratories Air Force Systems Command (AFWAL/FIBRA)
6c. ADDRESS (City, State and ZIP Code): North American Aircraft Operations (NAAO) El Segundo, California	7b. ADDRESS (City, State and ZIP Code): Wright-Patterson AFB, OH 45433
8a. NAME OF FUNDING/SPONSORING ORGANIZATION: Flight Dynamics Laboratory; 8b. OFFICE SYMBOL (If applicable): AFWAL/FIBRA	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER: F33615-81-C-3229
8c. ADDRESS (City, State and ZIP Code): Air Force Wright Aeronautical Laboratories Air Force Systems Command Wright-Patterson AFB, OH 45433	10. SOURCE OF FUNDING NOS.

PROGRAM ELEMENT NO.	PROJECT NO.	TASK NO.	WORK UNIT NO.
62201F	2401	02	49

11. TITLE (Include Security Classification) CADS - A Computer Aided Design System User's Guide (U)

12. PERSONAL AUTHOR(S) Less, Michael C., Manuel, Susan

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Yr., Mo., Day)	15. PAGE COUNT
FINAL	FROM 12/01/81 TO 4/30/85	85/08	225

16. SUPPLEMENTARY NOTATION

17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
FIELD	GROUP	SUB. GR.	Computer Aided Design, DI-3000, Finite Elements, NASTRAN, Structural Analysis & Optimization, Model Generation, Pre and Post Processor
01	03		
13	13		

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

This report is a user's guide for the Computer Aided Design System, "CADS." CADS is a pre and post processor for structural analysis and optimization programs based on the finite element method. The system supports five functional modules controlled by an Executive Monitor. All of these modules communicate with a data base through a data manager. In addition a post output translator, CADSPP, is available which processes output from finite element programs, e.g. NASTRAN, directly into the data base. This report gives a detailed description of all user commands available in CADS and CADSPP. A sample session is provided which shows the user commands applied to a variety of sample problems.

20. DISTRIBUTION/AVAILABILITY OF ABSTRACT: UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT. ☒ DTIC USERS ☐	21. ABSTRACT SECURITY CLASSIFICATION: UNCLASSIFIED
22a. NAME OF RESPONSIBLE INDIVIDUAL: V. A. TISCHLER	22b. TELEPHONE NUMBER (Include Area Code): 513-255-6992; 22c. OFFICE SYMBOL: AFWAL/FIBRA

DD FORM 1473, 83 APR EDITION OF 1 JAN 73 IS OBSOLETE.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

FOREWORD

This final report was prepared by Rockwell International, North American Aircraft Operations (NAAO), El Segundo, California for the Structures and Dynamics Division, Flight Dynamics Laboratory, (FDL) Wright-Patterson Air Force Base, Dayton, Ohio. The work was performed under Contract F33615-81-C-3229 which was initiated under Project No. 2401. Mrs. V. Tischler was FDL project engineer for this effort.

The "Development of a Computer-Aided Design System" (CADS) was a 41-month effort with this final report consisting of three volumes. Volume I, "Final Summary Report," presents an overview of the CADS software capabilities; Volume II, "User's Guide," contains the detailed instructions for each of the commands in the CADS software; and Volume III, "Program Maintenance Manual," describes the internal structure of CADS for use in future maintenance and enhancement of the code.

The Rockwell program manager for this effort was Mr. M. C. Less, NAAO Advanced Structures and Materials department. He was supported by Mrs. S. Manuel of the same department.

The work described in this report was initiated in December 1981 and completed in May 1985. This report was submitted for publication in May 1985.

Accesion For	
NTIS CRA&I	✓
DTIC TAB	☐
Unannounced	☐
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and / or Special
A-1	

TABLE OF CONTENTS

PREVIOUS PAGE IS BLANK

LIST OF ILLUSTRATIONS

LIST OF TABLES

1.0 INTRODUCTION

The widespread use of a large variety of finite element (FE) codes to perform structural analysis tasks has focused attention on a common Air Force and industry problem: the relatively large amount of time and effort required to perform data preparation, data validation, and resultant FE analysis tasks with existing state-of-the-art codes. This problem is further aggravated by the relatively slow, interactive response of mainframe time-sharing computer processing systems. To reduce time and effort, a computer-aided, advanced interactive graphics, minicomputer-based, finite element modeling system has been developed. This system includes mesh generation and validation capabilities as preprocessing functions as well as interactive graphic features for postprocessing the analysis code output data.

The Computer Aided Design System (CADS) software's most important aspects are that it is targeted for 32-bit minicomputer hardware, makes use of Fortran 77 and device independent graphics, and supports the definition of composite elements. The CADS program utilizes VAX 11/780 hardware with secondary testing for transportability, having been performed on IBM 4341 and PRIME 850 hardware. CADS is modular in nature with various functional modules accessed through a common Executive Monitor and makes use of common data base routines, as shown in Figure 1.

The contract, F33615-81-C-3229, was initiated in December 1981 and completed in May 1985 with these final reports. Volume I, "Final Summary Report," presents an overview of the CADS software capabilities; Volume II, "User's Guide," and Volume III, "Program Maintenance Manual," give detailed user instruction and source code descriptions of the CADS software. These three volumes make up the final documentation of the CADS software.

This User's Guide contains command descriptions and examples for all of the CADS capabilities. Each section of the manual covers a different aspect of the CADS software commands and their use.

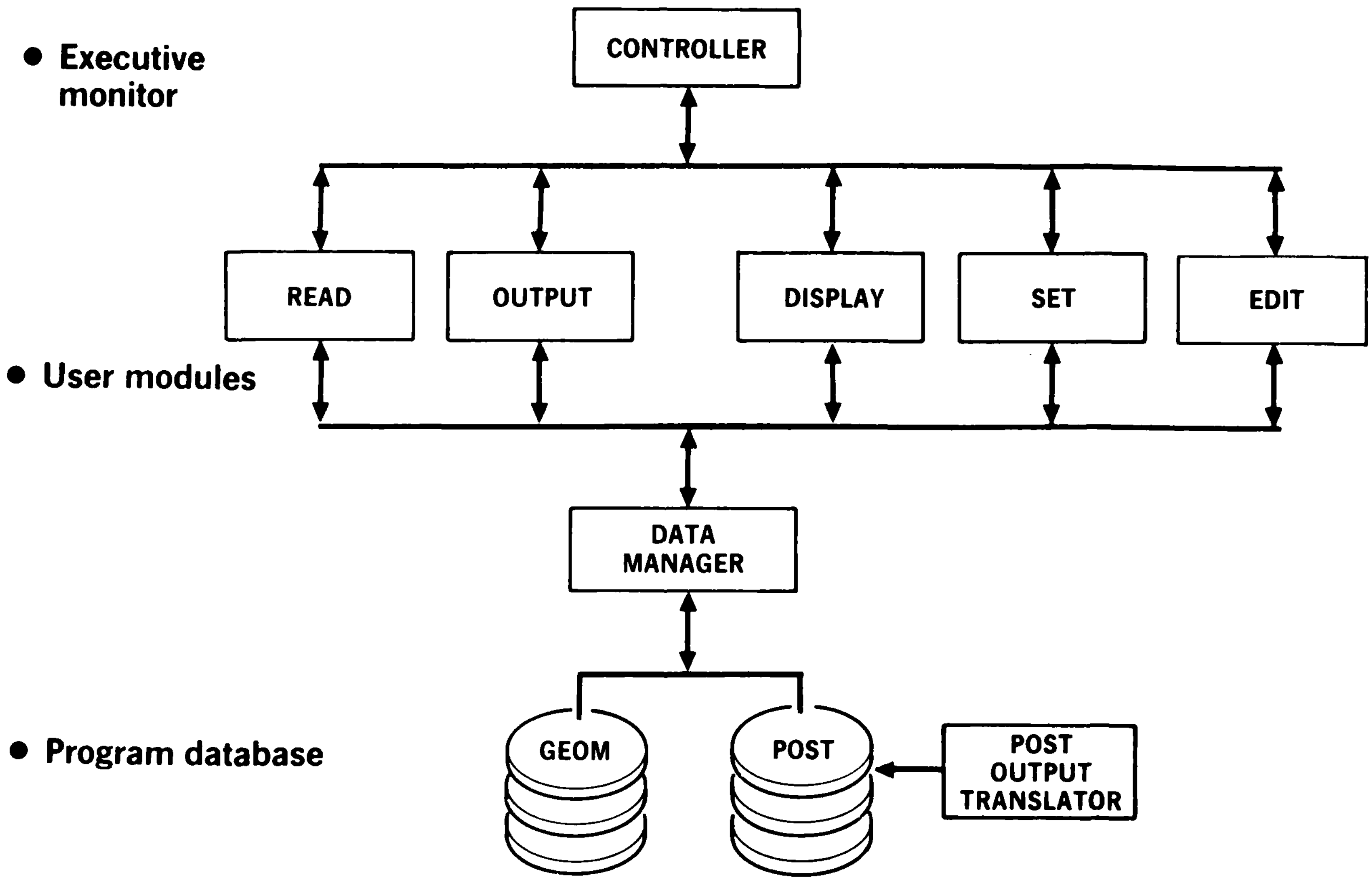


Figure 1. Modular Nature of CADS Software

2.0 GENERAL INFORMATION

2.1 COMMAND DRIVEN

CADS is a command-driven program which interprets and executes individual user commands. Commands are entered in free format using commas, blanks, or equal signs as delimiters. In general, all commands can be executed using a two character abbreviation - the first two letters of the command. However, in certain instances where there are multiple commands starting with the same characters, at least the first four characters of the word must be supplied. In decoding an input line the free read routine breaks each variable or entity in the command into an eight-character variable. Therefore, command line variables must be limited to eight characters or else the variable will be truncated to eight characters and a warning message will be printed on the display.

2.2 SPECIAL CHARACTERS

A number of special characters are available to simplify the input. These characters and their functions are:

$ - end of record symbol - used for comments
& - the continuation symbol - used for multiple line inputs
* - a multiplication symbol - used to repeat the same variable multiple times in a line.

More specifically, the $ sign ends the logical record so that the rest of the physical record can be used for comments. The & sign ends the current physical record and tells the free read routine to continue the logical record on the next physical record. The * sign is used to repeat a variable in a logical record. For example, the character string 5*10.0 would repeat the value 10.0 five times. It is equivalent to the string 10.0 10.0 10.0 10.0 10.0 and thus can save a significant amount of input typing. Finally, each logical record is limited to no more than 99 variables. Examples using each of these characters follow:

```
Example 1:   THIS IS A LINE OF VALUES $ NOW THIS IS COMMENT
Example 2:   VARIABLES GO BEYOND ONE LINE       &
             SO CONTINUE ON NEXT LINE           &
             FOR UP TO 99 VARIABLES
Example 3:   THE * REPEATS VARIABLES LIKE 3*VALUE &
             WOULD REPRESENT VALUE VALUE VALUE
```

2.3 LIST GENERATOR

On many commands a list of values may be specified for processing. A list has the following general format: N1 N2 N3 N4 TO N5 BY N6 where the N1,---,N6 are integer values and TO and BY are optional keywords. TO/BY acts to generate a list between N4 and N5 at N6 increments. List examples follow:

List Command		Members of List
5 10 11 9 7	=	5, 7, 9, 10, 11
1 4 28 TO 31	=	1, 4, 28, 29, 30, 31
28 TO 32 BY 2	=	28, 30, 32
1 5 TO 10 BY 2 17 20	=	1, 5, 7, 9, 17, 20
5 TO 8 10 TO 14 BY 2	=	5, 6, 7, 8, 10, 12, 14

2.4 MANUAL DEFINITIONS

The CADS program functions are broken into modules with processors acting under the modules. The program is controlled through the Executive Monitor which calls in the READ, OUTPUT, SET, DISPLAY, and EDIT modules. All of these modules communicate with the program's data base through a common set of input and output subroutines. Figure 2 shows the Executive Monitor and the relationship of the various modules and processors under it.

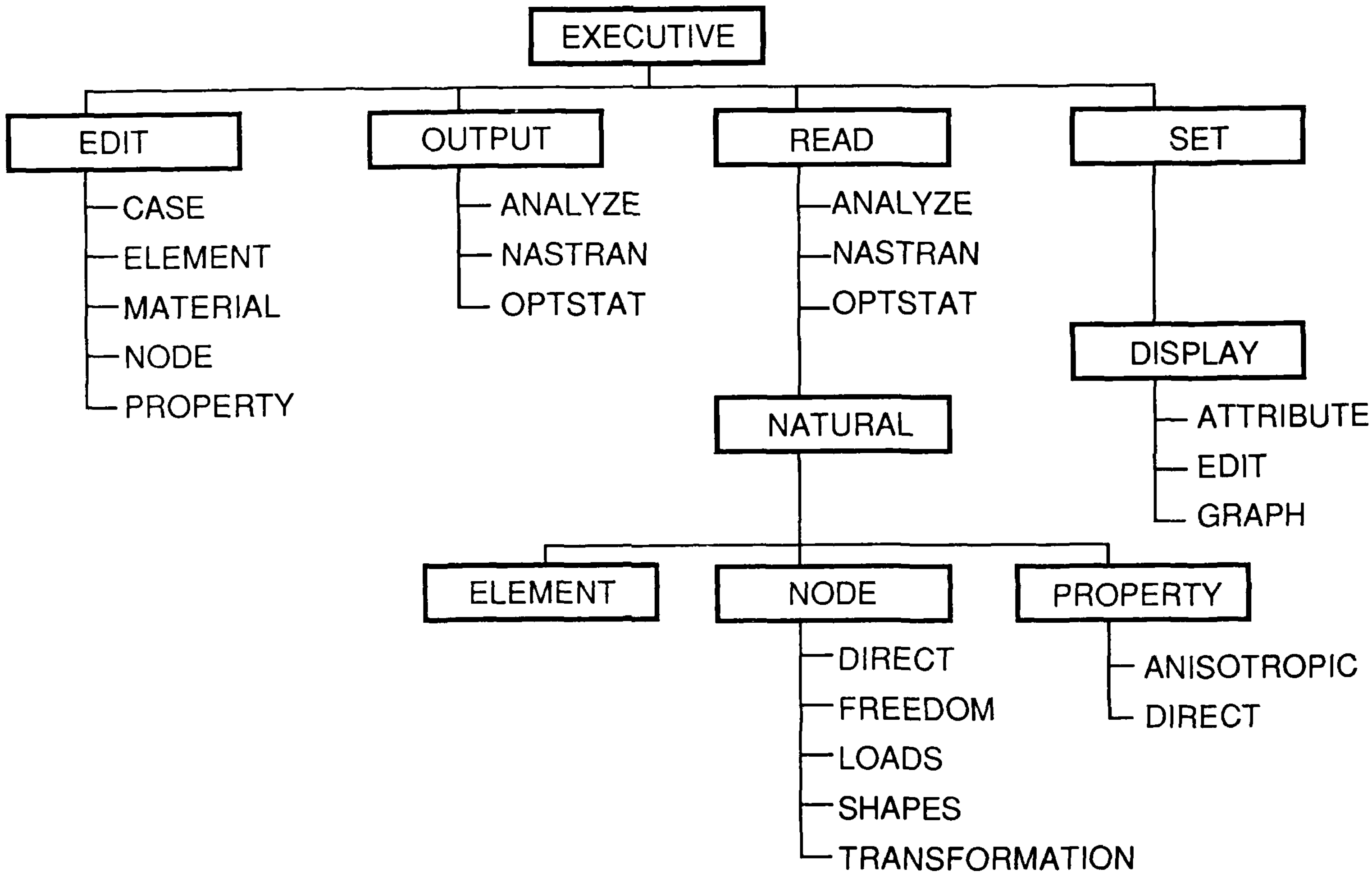


Figure 2. Overall Program Layout

3.0 EXECUTIVE MONITOR

3.1 INITIAL DIALOGUE

The Executive Monitor is used to establish the communication parameters for the program and to control the flow between program modules. After beginning program execution, described in section 10, the following dialogue will appear:

```
ENTER THE TERMINAL BEING USED.
VALID TYPES: ALPHA, 4014, CALC
```

where one of the following responses is required:

alpha:	for an alphanumeric terminal - no graphics
4014:	Tektronix 4014 graphics terminal
calc:	Calcomp plotter (not yet supported)

If the 4014 or another graphics terminal is being used, CADS will then ask for the communication or baud rate to be used. The following prompt is used:

```
ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ... 19200.
(THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
```

Valid responses are given as integer numbers in tens of characters per second. For example, dial-up lines are usually 300 or 1200 baud while direct lines will run at 4800, 9600, or 19200 baud. The actual baud rate to be entered will depend upon the terminal, communication lines and host computer and must be coordinated with the appropriate operating system personnel. An integer zero (0) is entered to request the default baud rate.

Once one of these responses is received, the program sets up internal switches and counts for the particular device type. It then prompts for the next input required by the program as follows:

ENTER THE PROGRAM COMMUNICATION TYPE:
(RESPOND EITHER: NASTRAN, ANALYZE, OPTSTAT OR NATURAL)
? START

valid responses are:

nastran:	input deck in NASTRAN format.
analyze:	input deck in ANALYZE format.
optstat:	input deck in OPTSTAT format.
natural:	input steering file in NATURAL generation format.

The communication type is used to specify the element naming convention to be used by the program in the SET, DISPLAY, and READ modules. Elements will be identified with the names used by the requested format. Since the communication mode sets the element naming convention it is independent of the type of data deck which may be read into CADS through the READ module. For example, if the NASTRAN communication mode is set then the shear panels are called CSHEAR, bending beams CBAR, and similarly for the other element types no matter what type of data deck is read in; i.e., NASTRAN, NATURAL, ANALYZE, or OPTSTAT. However, if the NATURAL mode was set then these elements are called out by their generation names which in the case of the shear and bending beam elements would be QS4 and B2, respectively. Three more start messages complete the CADS initialization.

DO YOU HAVE A POST PROCESSED ANALYSIS FILE (Y/N)?

where the valid responses are:

yes:	if a post data base is going to be used
no:	if a post data base will not be used

If a post data base is being used the following prompt is received:

ENTER POST DATA BASE FILE NAME FOR CADS PLOTTING OR END TO SKIP

the valid responses are:

end: to skip the allocation of a post data base
name: file name (max. 40 characters) for post data base

This name is then attached to unit 4 as a direct access post data base.The next question is for the permanent geometry data base used by CADS. It will be attached to unit 1. If a previously defined data base is not being used, CADS will generate a new data base for this execution of the program. The prompts for this are:

WILL YOU USE AN EXISTING DATA BASE (Y/N)?

valid responses are:

yes: an existing file will be used
no: a new geometry data base is to be generated

If yes is entered the prompt is:

ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP

where responses are:

end: to skip to the new data base generation
name: file name (max. 40 characters) for geometry data base

If a new data base is to be generated the following prompts are used:

ENTER THE TITLE FOR THE MODEL HEADER

the response is a model title of up to 72 characters.

The geometry data base name is defined using:

ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP

where the valid responses are:

end: to stop CADS execution

name: file name (max. 40 characters) for new geometry data base

The above dialogue determines the type of terminal used to execute the program and then the format in which input descriptions of the elements will be given. For example, the 4014 terminal type will probably be used most often since typically graphics displays will be needed. If the NASTRAN communication type is given, then axial rod elements will be called CROD; triangular membranes, CTRMEM; beams, CBAR; and so on for the remaining element types. The prompt string is ? NAME where NAME is the module name currently being executed. The file name questions set up the connections for using the direct access postprocessed analysis output data and/or a previously generated geometry file.

3.2 MODULE ACCESS

After these questions are answered, the executive monitor prompts for the next command using the prompt string ? CADS. At this point any of the valid program functional modules may be specified. The following are available:

read: reads in bulk data or model generation steering files

output: outputs data base information in a specified bulk data format

set: defines sets of nodes and/or elements for plotting and other functions

display: displays information at a graphics terminal through the SET module

edit: edits the current geometry data base and saves as a permanent file

end: terminates the CADS program and returns control to the host processor

These commands are input in free format with the first two characters being sufficient to define the option. In order to enter one of these modules, the module name is entered. This will begin execution of the selected module and allow execution of any of its commands. Note that the END command must always be completely spelled out.

4.0 READ MODULE

4.1 BEGIN READ COMMAND

The READ module is used to read a finite element model's data and to translate it to the CADS geometric data base. Basically, this involves the development of translator interfaces which are capable of decoding input bulk data information so that it can be stored in the data base through the data manager routines. The prompt string for this module is ? READ. The following commands are valid in the READ module:

BEGIN - used to start a translator processor
END - ends the READ module and returns to the Executive Monitor

The BEGIN command format is:

BEGIN processor INPUT unit FORMAT ggggeeee SPC NA list DISPLAY

The "processor" parameter is required since it defines the type of translation to be performed. The valid processor types are:

NASTRAN - for NASTRAN Bulk Data Decks
OPTSTAT - for OPTSTAT Data Decks
ANALYZE - for ANALYZE Data Decks
NATURAL - for model generation functions

The INPUT keyword defines the Fortran unit from which the program will read an input data deck. The default is unit 21; if another unit is used it must be specified as an integer number after the INPUT keyword. Only unit numbers greater than 21 should be used since units between 1 and 20 are assigned for various other program uses. The INPUT keyword is normally not used except when the NATURAL processor is being used with an input steering file. The standard procedure is to allow CADS to assign the unit numbers.

The FORMAT keyword is used for NASTRAN bulk data decks and breaks apart the eight-digit NASTRAN element number into groups and element offsets within the groups. The number of g's and e's specify the breakup procedure. For example, if three g's and five e's are given, then the first three characters in the NASTRAN element number will form the group number, while the second five numbers are used to determine the particular NASTRAN element's location or offset within the given group. Note that the g and e string must be one string without embedded blanks.

The SPC keyword is used with the NASTRAN processor. Its use will stop the normal processing of SPC1 NASTRAN bulk data cards and not update the constraints stored for the grids based upon the latest SPC1 set. By default the last set of SPC1 cards read in for the model will be stored as the valid constraints for the model's nodes.

The NA keyword is optional and is used to provide additional user control to the NASTRAN read module. The NA keyword is followed by a list of Fortran unit numbers containing additional data files for the NASTRAN processor. The NA keyword adds the information from the specified units to the current model as if it was a long data file. For example, the command

```
BEGIN  NASTRAN  INPUT = 21  NA = 22,23,24
```

would read information in NASTRAN format from units 21, 22, 23, and 24 as one long file. This feature may be used when the GRID data are in one file with the model's element, material, and load data contained in other data files. All input file names except for the first one must be preassigned before entering CADS.

The DISPLAY keyword is mutually exclusive with the FORMAT and INPUT keywords. It applies to the NATURAL processor and is used to tell the processor to display the model as it is generated at the terminal.

If the DISPLAY keyword is used, the following prompts will be used to request the maximum model sizes. This information is used to correctly scale the displays to the terminal. The prompts are:

DO YOU WISH TO PLOT NODES DURING GENERATION (Y/N)?

the valid responses are:

yes   nodes will be plotted during NATURAL generation
no: nodes will not be plotted during NATURAL generation

If yes is given the following prompt is received:

INPUT XMIN,XMAX YMIN,YMAX ZMIN,ZMAX

The response is six real numbers defining the x, y, and z axis limits.

After the BEGIN command is processed and decoded the following prompt is received in all cases except when the NATURAL INPUT=5 keywords were used in which case all input commands will come from the terminal. If not CADS assumes that an input card image file is to be attached and will then prompt for that file using:

ENTER program name INPUT FILE NAME NOW OR END TO RETURN

where the program name will be NASTRAN, ANALYZE, OPTSTAT or NATURAL as appropriate and the valid responses are:

end: to end the READ module and return to the Executive Monitor
name: card image file name (max. 40 characters) to be read into the geometry data base

As an example, the following series of commands would be used to read in an existing NASTRAN bulk data deck into CADS. The file name is NASTBULK.DAT.

```
?  CADS
   read
?  READ
   begin nastran
   ENTER NASTRAN INPUT FILE NAME NOW OR END TO RETURN
   nastbulk.dat
```

Other examples are provided in greater detail in section 12.0.

4.2 NATURAL GENERATION PROCESSOR

The NATURAL processor under the READ module provides the user with the capability to generate grids, elements, and element attributes for finite element models. Three submodules are available in the NATURAL processor: NODE, ELEMENT, and PROPERTY for generating grid data, element connectivities and element attributes.

The NODE submodule provides the means to generate node coordinates through direct, biased linear, repetition, and shape commands. It also provides the means for specifying grid suppressions and external loads. The ELEMENT submodule defines the element types and connectivities to be used for the model. The PROPERTY submodule is used to define element geometric and material properties for the model.

The DISPLAY keyword on the BEGIN NATURAL DISPLAY command will automatically display a model as it is generated at the terminal. When this keyword is used, minimum and maximum x, y, and z coordinate values are requested to scale the model to the terminal. After this is accomplished, the nodes generated in the DIRECT and SHAPES NODE subprocessors are automatically displayed as a generation command is entered. In the ELEMENT submodule the elements are displayed.

The syntax for the NATURAL processor is

```
BEGIN submodule
END
```

where BEGIN starts execution of the submodule and END ends the NATURAL processor and returns control to READ. The valid submodules are:

NODE: node data generation
ELEMENT: element connectivity data generation
PROPERTY: element attribute generation

The following commands are valid in the DIRECT and SHAPES subprocessors and the ELEMENT submodule.

```
PLOT  NODE
SAVE
ROTATE X=value Y=value Z=value
```

where the NODE keyword requests the node numbers for the display produced by the PLOT command. The SAVE command saves the generated model data to the geometric data base and clears the screen. For the ROTATE command the X, Y, and Z keywords are used to specify the axis about which the given rotation, in degrees, will be performed.

4.3 NODE SUBMODULE

NODE controls various subprocessors to generate coordinates, apply boundary conditions, and develop external load data. The syntax is:

```
BEGIN    subname
END
```

where the subname is the name of the subprocessor to be executed and END ends the NODE module. The valid subnames are:

DIRECT: directly defines node coordinates
SHAPES: defines nodes along curved shapes
FREEDOM: specifies node boundary conditions
TRANSFORM: transforms selected node locations
LOADS: defines external loads

4.3.1 DIRECT NODE SUBPROCESSOR

The DIRECT subprocessor provides commands for coordinate definitions on a point-by-point, a linear interpolation, a biased line, and a repetition factor basis. The syntax for the DIRECT subprocessor is as follows:

```
PLOT  NODE
SAVE
ROTATE  X=value  Y=value  Z=value
AXIS x y z
MIRROR axis increment
MIRROR OFF
CENTER axis increment
CENTER OFF
PERCENT sum factor 1....factor n
PERCENT OFF
LIST NODE  list
LIST GROUP list
EQUATE list
NODE n1 x1 y1 z1 TO n2 x2 y2 z2 BY n3
NODE n1 x1 y1 z1 TO n2 BY n3 ALIGN n4 DELTA d1
REPEAT n n1 x1 y1 z1 n2 x2 y2 z2 n3
END
```

The PLOT, SAVE, and ROTATE commands act as described in section 4.2.

The AXIS command allows the user to specify the order of the coordinate data input with respect to the axes and, more importantly, to drop an axis if the problem is two-dimensional. For example, the command

```
AXIS X Y
```

would say that the first coordinate value is the x value, the second is y, and all z values will be zero and not specified. Once specified it will remain on until changed. The default is AXIS X Y Z.

The MIRROR command mirrors the generated nodes about the specified axis or axes incrementing the node number by the given value. Once turned on, the command remains on until turned off. The command line

MIRROR Y Z 100

would generate an additional node with a number N + 100 located at x, -y, and -z of the original node for each generated node N. Mirror OFF will turn off the MIRROR command.

The CENTER command is basically the same as the MIRROR command with the exception that CENTER sets the given axis coordinate to 0.0 instead of mirroring it. CENTER OFF will turn off the CENTER command.

The PERCENT command is used to bias the default, equally spaced interpolation process, to a user defined percent process. The command requires a value and list of factors which sum to the first value. The factors are basically the percent distances between the nodes on a given line. The command determines the percent locations of the factors with respect to the given sum and places nodes along a new line at those locations. For example,

PERCENT 100.0 10. 20. 20. 40. 10.

will define percent locations of 0, 10, 30, 50, 90, and 100 percent for a series of six nodes. Typically, 100 and percentages of 100 are used for the command, but any value and factors summing to that value can be used. For instance, the sum value could be 12.5 representing a distance which is factored into a list of values: 1.0, 2.5, 1.5, 3.0, 2.0, 2.5. These factors sum to 12.5 and would yield nodes at 0.0, 1.0, 3.5, 5.0, 8.0, 10.0, and 12.5. The command for this example is:

PERCENT 12.5 1.0 2.5 1.5 3.0 2.0 2.5

The defined percentages are used for all new lines until PERCENT is set to OFF.

The LIST command will list (at the terminal) all of the nodes or element groups requested by the "input list." The input list may be blank or a list of integers or a TO/BY list, as discussed in section 2.0 of this manual. The NODE keyword delineates a list of nodes to be printed while GROUP specifies a list of element groups to be printed. If a blank string is entered, then all of the model's nodes or groups will be printed. For example, the command cited below will list the nodes and coordinates for all nodes between 1 and 10.

```
LIST NODE 1 TO 10
```

The EQUATE command provides the user with the capability to change a previously defined node number to a new number. It's syntax is the word EQUATE followed by up to 48 sets of new and old node numbers. This command takes the second node number of a pair and replaces it with the first or new number of the pair. The syntax is

```
EQUATE  list
```

where the list has the sets of new and old node numbers to be equated. For instance,

```
EQUATE  201   1   202   2   203   3
```

would change the node numbers for 1, 2, and 3 to numbers 201, 202, and 203.

The NODE command is used to specify the node numbers and the coordinates of nodes to be generated along a line in space. By default, equal increments are used to space the generated nodes. If a PERCENT command has been specified, the nodes would be biased by those percent values. The syntax for the NODE command is:

```
NODE n1 x1 y1 z1 TO n2 x2 y2 z2 BY n3
NODE n1 x1 y1 z1 TO n2 BY n3 ALIGN n4 DELTA d1
```

where n1, n2, and n4 are node numbers, n3 is an increment, d1 is a delta distance, and the x, y, and z's are the coordinates of the two end nodes (line end points). The first node format says to generate a line of nodes starting at node n1, extending to node n2 and incrementing the node numbers between n1 and n2 by the n3 value. For example,

```
NODE 101 0.0 1.0 2.0 TO 111 0.0 5.0 0.0 BY 2
```

will generate six nodes numbered 101, 103, 105, 107, 109, and 111 starting at (0.0, 1.0, 2.0) and equally spaced to (0.0, 5.0, 0.0). This command defaults to an increment value of 1. Single points may be generated by specifying the node number and its location; if a node has already been defined, it can be used as a start or end point without respecifying its coordinates. For example, the two commands

```
NO 1 0.0 1.0 0.0
NO 1 TO 11 0.0 5.0 0.0
```

will generate ten new nodes numbered (2 through 11) along a line between node 1 at (0.0, 1.0, 0.0) and node 11 at (0.0, 5.0, 0.0).

The ALIGN and DELTA keywords are used with the NODE command to provide the user the capability to generate nodes along a straight line defined between the first node number on the NODE card and the node given after the ALIGN keyword. Nodes are then generated along this line at equal intervals as given by the DELTA keyword until the addition of another delta increment would place a new node beyond the node given by the ALIGN keyword. For example, the commands

```
NODE   10    23.0   15.0   2.0
NODE   101     0.     0.     0.  TO   111   BY   2   ALIGN   10   DELTA   5.0
```

will generate nodes 101, 103, 105, 107, 109, and 111 on a line from 0., 0., 0. to 23.0, 15.0, 2.0 at a spacing of 5.0 between the nodes. If the delta value had been greater than 5.5 and less than 6.9, only nodes 101, 103, 105, 107, and 109 would be generated since that delta value range will use up the distance between nodes 101 and 111 in four nodes. Essentially the ALIGN and DELTA keywords are used to place an integer number of nodes along a line from the first node to the ALIGN node. Instead of dividing the distance between those nodes into equal increments it is divided into an integer number of DELTA value increments.

The REPEAT command is very powerful since it allows the user to repeat the previous NODE command to generate repetitively located nodes, i.e., for plates, wing covers, etc. The REPEAT syntax is

```
REPEAT n nl x1 y1 z1 n2 x2 y2 z2 n3
```

The n is the number of times the previous node card is repeated using nl, n2,n3 and x,y,z values as the increments to be added to the corresponding NODE command values to generate another line of nodes. For example, the commands

```
NODE       1  0.0  0.0  0.0  TO    9  0.0  5.0  0.0  BY 2
REPEAT 4  10  1.0  0.0  0.0       10  1.0  0.0  0.0     0
```

will generate odd numbered nodes from 1 to 49 in five lines of five nodes. These two commands are equivalent to the following five commands:

```
NODE   1  0.0 0.0 0.0  TO   9 0.0 5.0 0.0 BY 2
NODE  11  1.0 0.0 0.0  TO  19 1.0 5.0 0.0 BY 2
NODE  21  2.0 0.0 0.0  TO  29 2.0 5.0 0.0 BY 2
NODE  31  3.0 0.0 0.0  TO  39 3.0 5.0 0.0 BY 2
NODE  41  4.0 0.0 0.0  TO  49 4.0 5.0 0.0 BY 2
```

The END command returns control to the NODE submodule.

4.3.2 SHAPES NODE SUBPROCESSOR

The SHAPES subprocessor under the NODES submodule allows the user to generate nodes along circles, ellipses, and arcs of parabolas. In each case the user basically provides the information to determine the correct equation for the command. For each command the nodes are generated counterclockwise unless the REVERSE keyword is used to cause the generation in the clockwise direction. The CIRCLE command is described by a CENTER, defaulted to x = 0.0, y = 0.0, and START location where the first node is assigned. The ELLIPSE command is described by a CENTER, START, MAJOR axis and MINOR axis defaulted to a circle of radius 1.0 centered at x = 0.0 and y = 0.0. The PARABOLA command is described by a FOCUS defaulted to x = 0.0 and y = 1.0, a START, and a VERTEX.

All three commands require a list of nodes to define the number of nodes and their grid point numbers to be generated. These nodes are defined at equal intervals through 360 degrees for the CIRCLE and ELLIPSE and through 180 degrees for the PARABOLA. In addition, the PERCENT command may be used to specify the spacing of the nodes along the circle, ellipse or parabola. This command is described by an angle and a series of deltas whose sum equals the given angle. The PERCENT command, therefore, allows the user to define nodes only along a given portion of a curve. With this command the START location is taken as the first point with the line from the center to the point as the 0 degree line so that the deltas are added from the START point through the given total angle. The number of nodes in the node list must be one more than the number of factors in a PERCENT command since the node list includes the start and end node. If a PERCENT command is used, it must immediately precede the appropriate SHAPES command. The keyword Z may be used with any of the commands to change the z location of the grid points from 0.0 to a specified value. The keyword GRIDS may be used instead of NODES to obtain the x, y, and z locations of the starting point instead of specifying them with the START and Z keywords.

The PLOT, SAVE, and ROTATE commands perform as described in section 4.2.

The commands of the SHAPES subprocessor are

```
PERCENT phi dang1 dang2 dang3 ...

CIRCLE CENTER = x1,y1 START = x2,y2 Z = z1 REVERSE
     NODES/GRIDS n1 TO n2 BY n3

ELLIPSE CENTER = x1,y1 START = x2,y2 Z = z1 REVERSE
     MAJOR = x3,y3 MINOR = x4,y4
     NODES/GRIDS n1 TO n2 BY n3

PARABOLA FOCUS = x1,y1 START = x2,y2 Z = z1 VERTEX = x3,y3
     REVERSE
     NODES/GRIDS n1 TO n2 BY n3

PLOT NODE

SAVE

ROTATE X = value  Y = value  Z = value

END
```

More specific command descriptions are:

PERCENT: This command provides the capability to generate the node locations at unequal intervals of arc or subtended angle. If used it must precede the command to which it applies. The first value is the total angle which is encompassed while the remaining values are the delta angles at which the nodes are to be placed along the curve. By defining a total angle of less than 360 degrees, a portion or arc of a circle, an ellipse, or a parabola can be defined.

CIRCLE: This command provides the user with a capability to create nodes along the arc of a circle. The nodes are generated at equal intervals in a counterclockwise direction. The keywords are:

CENTER	-	Specifies the center location at x1,y1
START	-	Specifies the starting position at x2,y2
Z	-	Specifies the z coordinate for the generated nodes if different from 0.0
REVERSE	-	Indicates nodes are to be generated clockwise
NODES/GRIDS	-	Specifies the node numbers to be assigned along the circle

ELLISPE: This command provides the user with a capability to generate nodes along the arc of an ellipse. The nodes are generated counterclockwise at equal intervals along the curve (delta subtended angle) unless otherwise specified. The keywords are:

CENTER	-	Specifies the center location at x1,y1
START	-	Specifies the starting position at x2,y2
Z	-	Specifies the z coordinate for the generated nodes if different from 0.0
REVERSE	-	Indicates that the nodes are to be generated clockwise
MAJOR	-	Specifies the major axis of the ellipse at x3,y3
MINOR	-	Specifies the minor axis of the ellipse at x4,y4
NODES/GRIDS	-	Specifies the node numbers to be assigned along the ellipse

PARABOLA: This command provides the user with a capability to create nodes along the arc of a parabola. The keywords are:

FOCUS	-	Specifies the parabola′focus at x1,y1
START	-	Specifies the starting position at x2,y2
Z	-	Specifies the z coordinate for the new nodes if different from 0.0
VERTEX	-	Specifies the parabola vertex at x3,y3
REVERSE	-	Indicates that the nodes are to be generated clockwise
NODES/GRIDS	-	Specifies the node numbers to be assigned along the parabola

Examples of these commands are given in Figures 3, 4, and 5. The command in Figure 3 will generate a complete circle from nodes 101 to 120 as shown. The ellipse command is shown in Figure 4 using nodes 201 to 220. A parabola is defined by the command in Figure 5. It is a full parabola and defines nodes 301 through 320.

The END command returns control to the NODE submodule.

4.3.3 FREEDOM NODE SUBPROCESSOR

The function of the FREEDOM subprocessor is to provide the user with a means for interactively changing the constraints of a structural model.

The nodes created by the NODES module are without constraint and are free to both translate and rotate. To prevent this motion from occurring, a node can be restrained using the SUPPRESS command; it should be restrained to satisfy the equations of equilibrium. The symbols TX, TY, and TZ refer to translational motion along the X, Y, and Z axes, respectively. The symbols RX, RY, and RZ refer to rotations about the X, Y, and Z axes, respectively. Note that in addition to the ability to restrain nodes, nodes which are incorrectly suppressed may be released to allow motion.

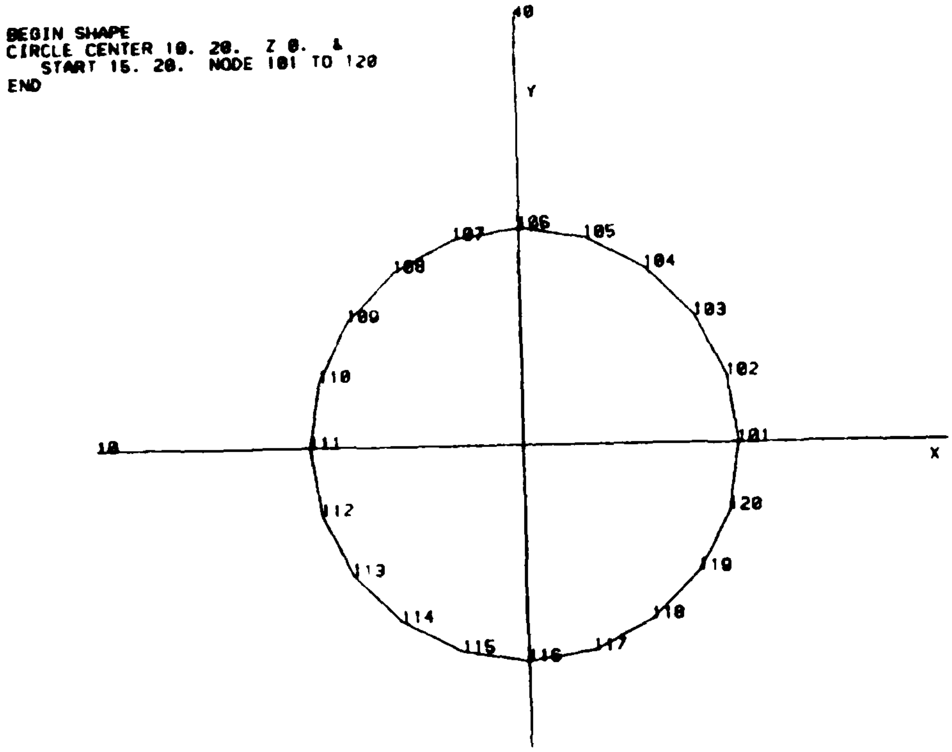


Figure 3. Example of Circle Shape

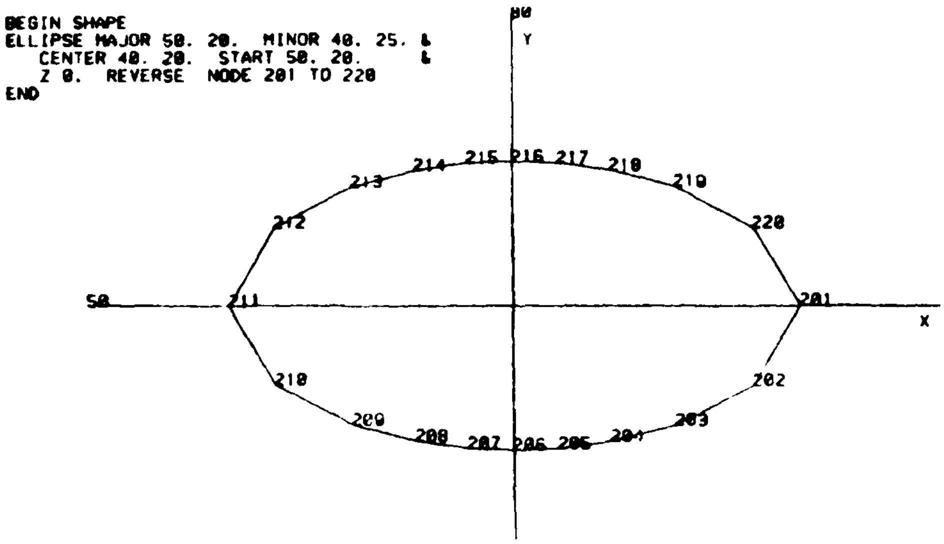


Figure 4. Example of Ellipse Shape

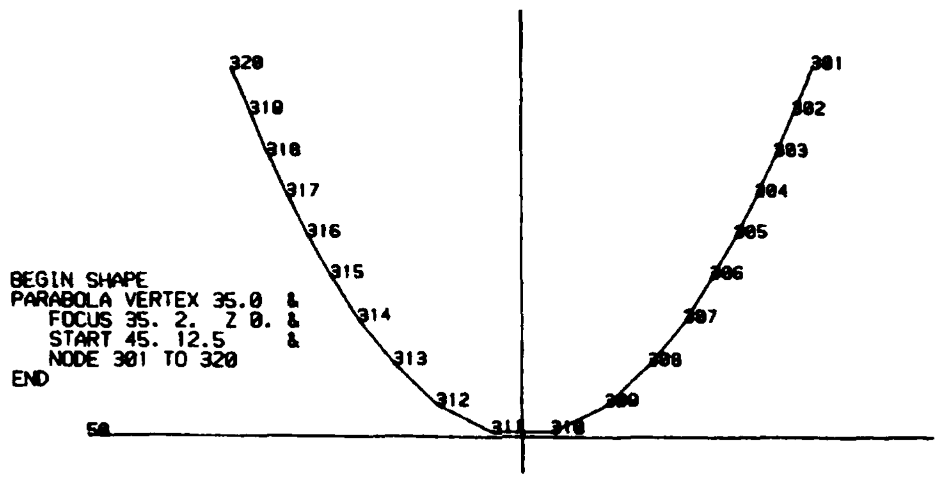


Figure 5. Example of the Parabola Shape

The commands for the FREEDOM subprocessor are:

SUPPRESS TX TY TZ RX RY RZ ALL SET N--- NODES n1 TO n2 BY n3
SFREE TX TY TZ RX RY RZ ALL SET N--- NODES n1 TO n2 BY n3
END

The SUPPRESS command identifies nodes to be altered in any of three ways: by ALL, by SET, or by list. The keyword ALL implies that all of the nodes in the entire structure are to be handled in the requested manner. The keyword SET followed by a node set name implies that all of the nodes in the requested node set are to be changed in the given manner. Note that the node set must previously have been defined by the commands in the SET module. The list option, following the keyword NODES, allows the user to select, by list generated means, a series of nodes to be altered in the specified manner.

For example, the following commands will suppress the three rotational degrees of freedom (RX, RY, RZ) on ALL of the nodes; next, suppress the translational (TZ) degree of freedom in the z direction on node set NS2, and then, suppress the three translational degrees of freedom (TX, TY, TZ) for every tenth node from 100 through 200.

```
SUPPRESS    RX  RY  RZ  ALL
SUPPRESS    TZ          SET   NS2
SUPPRESS    TX  TY  TZ  NODES 100 TO 200 BY 10
```

The SFREE command frees a previously defined suppression from the given nodes. These nodes may be defined as ALL, a node set, or a list. It may also be easier to define suppressions on a large group of nodes and then go back to selectively free some of those nodes. For instance, the following two commands would first suppress RX, RY, and RZ, the rotational degrees of freedom, for nodes 1001 through 1200 and then free RZ for odd nodes from 1101 through 1150.

```
SUPPRESS    RX  RY  RZ  NODES 1001 TO 1200
SFREE       RZ          NODES 1101 TO 1150 BY 2
```

The END command returns control to the NODES submodule.

4.3.4 TRANSFORMATION NODE SUBPROCESSOR

The function of the TRANSFORM subprocessor is to transform the coordinates of a user-defined local system to the model's global system. This may be accomplished in several ways and the user may transform several sets of nodes using different transformations.

The TRANSFORM subprocessor commands are:

```
OFFSET   X=value    Y=value  Z=value  ALL
                                      NODES list
                                      SET   name

ROTATE   X=value    Y=value  Z=value  ALL
                                      NODES list
                                      SET name
```

```
NODES    ORIGIN  n1   XAXIS n2 XZPLANE  n3    ALL
                      YAXIS    YZPLANE        NODES list
                      ZAXIS    XYPLANE        SET   name

POINTS   ORIGIN  x1,y1,z1  XAXIS  x2,y2,z2   XZPLANE x3,y3,z3  ALL
                           YAXIS             YZPLANE           NODES list
                           ZAXIS             XYPLANE           SET   name

CONNECT  xL1,yL1,zL1    xL2,yL2,zL2     xL3,yL3,zL3            ALL
         xG1,yG1,zG1    xG2,yG2,zG2     xG3,yG3,zg3            NODES list
                                                               SET   name

STRETCH  xL1,yL1,zL1    xL2,yL2,zL2     xL3,yL3,zL3            ALL
         xG1,yG1,zG1    xG2,yG2,zG2     xG3,yG3,zG3            NODES list
                                                               SET   name
END
```

To translate a coordinate system, the user enters the command OFFSET, followed by the X, Y, and Z values required to move the local system origin to the global system origin in terms of the global system coordinates. The user specifies the axes to be translated as shown:

```
OFFSET  X=value  Y=value  Z=value
```

If a rotation of the local system is required to align it to the global system, the user uses the ROTATE command followed by the X, Y, and Z rotational angles. The following example will rotate about the X, Y, and Z axes by 35.0, 40.0 and 60.0 degrees, respectively.

```
ROTATE  X=35.0  Y=40.0  Z=60.0
```

Note: If both OFFSET and ROTATE are specified on the same card, the translation or OFFSET will always be done after the axes are rotated.

By invoking the NODES or POINTS option, the user can allow the module to compute the necessary translations and/or rotations. To accomplish this, the user must supply three nodes for the NODES option or the coordinates of three points for the POINTS option in the global system such that

> One point corresponds to the origin of the local system
> One point lies on an axis in the local system
> One point lies in a plane containing the above axis and another axis (XZ, YZ, or XY plane), or it lies on another axis

These specified points must lie on the positive half of the axis or, if a plane is specified, in the quadrant containing the positive halves of two axes. To specify the origin, the user enters the keyword ORIGIN, followed by the node number of the point corresponding to the origin (for the NODES option), or the keyword ORIGIN, followed by the X, Y, and Z coordinates of the point (for the POINTS option). To specify an axis, the user enters one of the keywords XAXIS, YAXIS, or ZAXIS followed by the node number or coordinates of a point that lies on the positive half of the axis. To specify a plane, the user enters one of the keywords XYPLANE, XZPLANE, or YZPLANE followed by the node number or coordinates of the point that lies in the positive quadrant of the plane. The order in which the origin, axis, and plane are specified is not important. If node 1 had coordinates 1.0, 0.0, -3.0, node 7 had coordinates 1.0, 4.0, 9.0, and node 9 had coordinates 7.0, 3.0, 2.0, the following would be equivalent.

```
NODES   ORIGIN 9  XZPLANE 1  ZAXIS 7
POINTS   XZPLANE   1.0   0.0   -3.0 ZAXIS 1.0 4.0 9.0 ORIGIN 7.0 3.0 2.0
```

If the user knows the relationship between three points in the local system and three points in the global system, the module will compute the necessary translations and rotations needed to align the three points in the local system to the three points in the global system. All of the remaining points in the local system will be transformed using the same relationship. To accomplish this, the user enters the command CONNECT, followed by the coordinates of three points in the local system, and then the corresponding coordinates in the global system. The user must insure that the points in the local system are congruent to those in the global system. That is, the points must be oriented in the same way with the same distances between them. Further, the three points cannot be colinear.

If the user wishes to induce a distortion into the transformation, the STRETCH keyword may be used. The STRETCH option is identical to the CONNECT option, except that no checks are made to determine if the points specified for the local system are congruent to the points specified for the global system. The points cannot be colinear.

The nodes constituting the local system may be specified in one of three ways. By entering the keyword ALL, all of the nodes thus far defined will be transformed: this is the default option. By entering the keyword SET, followed by a node set name, a specific set will be transformed. By entering the keyword NODES, a node list can be specified.

The END command returns control to the NODES submodule.

4.3.5 LOADS NODE SUBPROCESSOR

The function of the LOADS subprocessor is to provide the user with the means to interactively apply external forces to a set of nodes. When nodes are defined no applied load information is defined for them. This subprocessor takes the user inputs for case number, reference coordinate system, scale factor, load direction, and applicable nodes to make up a set of externally applied loads. These loads are then output, in the OUTPUT module, as forces for the specified analysis program.

The commands for the LOADS subprocessor are:

```
CASE      number
FL        value
FM        value
CID       number
LOAD      X=value   Y=value   Z=value   NODE   list
MOMENT    X=value   Y=value   Z=value   NODE   list
END
```

The CASE command identifies the case or condition number for the load or moment commands which come after it. The FL and FM commands are the scale factors for the applied LOAD or MOMENT commands. The CID is the coordinate system reference number for the applied loads or moments. It defaults to the base system and is only applicable to NASTRAN output. Once defined the CASE, FL, FM, and CID commands remain at those values until specifically redefined by the user.

The LOAD and MOMENT commands define the translational force (LOADS) or rotational moment (MOMENT) vector directions to be applied to the given list of NODES. The X, Y, and Z keyword values are real numbers which define the direction of the applied vector. Essentially the FL or FM factor is multiplied by the X, Y, and Z component values to obtain the actual applied force or moment. This force or moment is then applied to each node in the given node list. The list is the standard TO/BY type of list used throughout the program. It should be noted that only NASTRAN can process applied moment values. No conversion of moments is attempted for OPTSTAT or ANALYZE.

For example the commands:

```
CASE    1001
FL      500.0
LOAD  X=1.0  Y=1.0  NODES  1 TO 50
```

will define a load case numbered 1001 with an applied force of 500.0 units in the x and y directions on nodes 1 through 50.

4.4 ELEMENT GENERATION SUBMODULE

The ELEMENT submodule provides the means for generating element types, groups, and connectivities for a wide variety of different elements. There are ten commands in the ELEMENT submodule:

```
GROUP number

DUPE GROUP n INCREMENT n

LIST GROUP list

LIST NODE list

TYPE nodes1 TO nodes2 BY inc CLOSE nodes

REPEAT n nodes1 nodes2 inc

SAVE

ROTATE X = value  Y = value  Z = value

PLOT   NODE

END
```

The GROUP command specifies the group number to be used for the elements defined by the commands which follow it. The group philosophy provides an easy and powerful means for splitting a structural model into smaller, more easily displayed and managed sets of elements. The group's only restriction is that each group must be composed of only one element type. The GROUP command is used to separate out the generated elements. After a GROUP command, the DUPE, TYPE, and REPEAT commands define the element types and connectivities used to define the group.

The DUPE command duplicates a previously defined group incrementing the connectivities of the first group by assigning an increment value to define the connectivities of the new group. For example, the commands

```
GROUP 5
DUPE GROUP 2 INC 100
```

will generate a group 5 of elements of the same type and connectivity patterns as group 2. The group 2 nodes would be incremented by 100 to obtain the group 5 nodes. This command is generally used in generating the lower cover of a wing after the upper cover is defined.

The LIST GROUP and LIST NODE commands will list the groups or nodes previously defined. The list of groups or nodes to be printed follows the GROUP or NODE keywords. The LIST command is described in the NODE submodule in section 4.3.1.

The TYPE command is used to specify the element type and its node connectivities. This command allows the generation of a string of elements using the TO and BY keywords which generate elements from Nodes 1 TO Nodes 2 by increments. Table 1 defines the various element type generation names, their NASTRAN types, and their NASTRAN and CADS property types. In all TYPE commands, the word TYPE is replaced by the appropriate element type name from Table 1.

For example, a series of axial rod elements along a spar cap (defined by the odd nodes 1 through 49) would be entered as

```
CROD 1 3 TO 47 49 BY 2 2
```

This command will generate CROD NASTRAN type elements between nodes 1 and 3, 3 and 5, and so on through nodes 47 and 49. Four rows of quadrilateral elements for a wing cover defined by the five rows of odd numbered nodes 1-21, 101-121, 201-221, 301-321, and 401-421 would be generated by:

TABLE 1

ELEMENT TYPES SUPPORTED BY CADS

ANALYZE OPTSTAT		CADS ELEMENT		NASTRAN ELEMENT	CADS PROPERTY	NASTRAN PROPERTY
2	=	CROD	=	CROD	PR2	PROD
2	=	R2	=	CONROD	PR2	
		B2	=	CBAR	PB2	PBAR
3	=	TM	=	CTRMEM	PID	PTRMEM
4	=	QM1	=	CQDMEM1	PID	PQDMEM1
		TB2	=	CTRIA2	PID	PTRIA2
		QB2	=	CQUAD2	PID	PQUAD2
		TB1	=	CTRIA1	PTQ1	PTRIA1
		QB1	=	CQUAD1	PTQ1	PQUAD1
		RQ4	=	CTRAPAX	PRTQ	PTRAPAX
		RT3	=	CTRIAAX	PRTQ	PTRIAAX
5	=	QS4	=	CSHEAR	PID	PSHEAR
		QT4	=	CTWIST	PID	PTWIST
		B2A	=	CPIPE1	PB2A	PPIPE1
		TM6	=	CTRIM6	PTM6	PTRIM6
		QM8		----	PQM8	
		SO4	=	CTETRA		
		SO6	=	CWEDGE		
		SO8	=	CIHEX1	PS82	PIHEX
		SO20	=	CIHEX2	PS82	PIHEX
		AS	=	CELAS1	PAS	PELAS
		TB3	=	CTRIA3	PSHE	PSHELL
		QB4	=	CQUAD4	PSHE	PSHELL

NOTE: The nodes are defined in the same order as NASTRAN defines the nodes for these elements.

```
QM1    1    3  103  101  TO   19   21  121  119  BY  4*2
QM1  101  103  203  201  TO  119  121  221  219  BY  4*2
QM1  201  203  303  301  TO  219  221  321  319  BY  4*2
QM1  301  303  403  401  TO  319  321  421  419  BY  4*2
```

This illustrates the BY keyword which defines the increments between corresponding node positions. The * multiplier special character repeats the 2 four times so that each node position, corresponding to the 1, 3, 103, and 101 is incremented by 2 until the 19, 21, 121, and 119 nodes are reached.

The CLOSE keyword of the TYPE command provides a method for automatically closing a string of elements, for example, a circular section of some part. The command:

```
QM1 1  2  402  401 TO 9  10  410  409  BY 4*1 CLOSE 10 1  401  410
```

will generate ten quadrilateral membrane (CQDMEM1) elements with connectivities from 1, 2, 402, 401 through 9, 10, 410, 409 with increments of 2 on the nodes and a final membrane between 10, 1, 401, 410. An example is shown in Figure 6.

The REPEAT command is very similar to the REPEAT command under the NODE submodule except that, in the ELEMENT submodule, it repeats on the previously defined element type card and its node connectivity definition. For example, the sets of commands

```
R2     1  2   TO   9  10  2 2
RE 4  10 10       10  10  0 0
```

and

```
R2   1   2  TO   9 10 BY 2 2
R2  11  12  TO  19 20 BY 2 2
R2  21  22  TO  29 30 BY 2 2
R2  31  32  TO  39 40 BY 2 2
R2  41  42  TO  49 50 BY 2 2
```

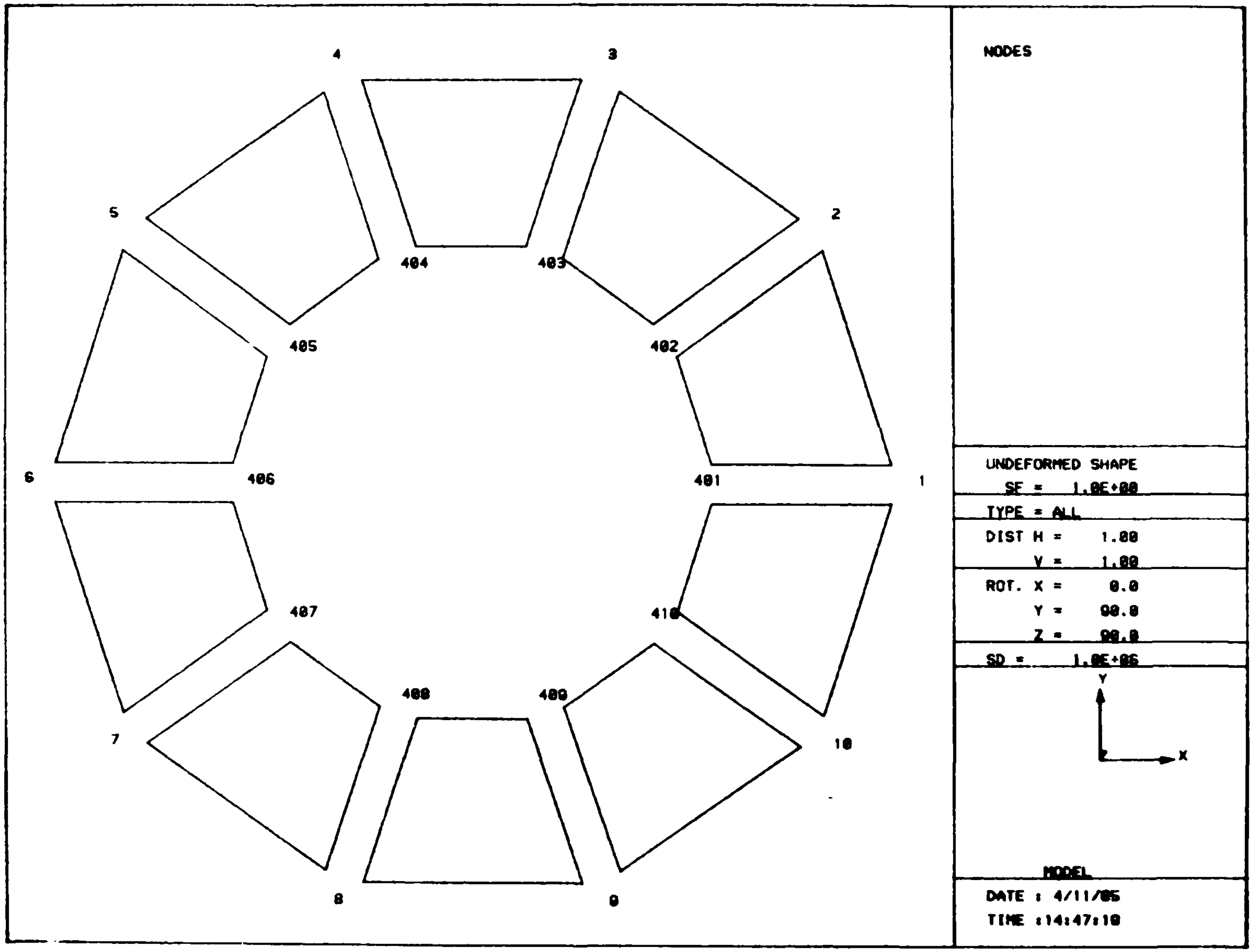


Figure 6. Circular Section - Ten Quadrilateral Elements

will (in both cases) generate five sets of five axial rods between nodes 1-2, 3-4, 5-6, 7-8, 9-10, etc.

The END command ends the ELEMENT submodule and returns command to the NATURAL module.

4.5 PROPERTY GENERATION SUBMODULE

The PROPERTY submodule is used to specify element sizes and materials for the particular structural analysis problem. The syntax is:

```
BEGIN subprocessor
END
```

where BEGIN starts a subprocessor and END ends the PROPERTY submodule returning control to the NATURAL processor. DIRECT and ANISOTROPIC are the valid subprocessors for the PROPERTY submodule. The DIRECT subprocessor is for defining element materials and thickness attributes directly to each group of elements. The ANISOTROPIC subprocessor provides commands for defining layered composite property values.

4.5.1 DIRECT PROPERTY SUBPROCESSOR

The DIRECT subprocessor is used to define the material and size tables to be assigned to the different element groups. Four types of material tables, corresponding to the MAT1, MAT2, MAT4, and MAT5 NASTRAN material types, may be defined. Several different size tables are supported based on the data definition requirements for the various element types which are supported. In all cases, the basic format for the commands is the same: a command name followed by a table identification number followed by a series of keywords and property or size values. The prompt string is ? DIRECT. The valid material commands are given below:

```
MAT1  id  E     v1   G    v2   U    v3   RHO  v4   A    v5
          TREF  v6   GE   v7   ST   v8   SC   v9   SS   v10

MAT2  id  A11   v1   A12  v2   A13  v3   A22  v4   A23  v5   A33  v6
          RHO   v7   A1   v8   A2   v9   A3   v10  TREF v11
          GE    v12  ST   v13  SC   v14  SS   v15

MAT4  id  K     v1   CP   v2

MAT5  id  KXX   v1   KXY  v2   KXZ  v3   KYY  v4   KYZ  v5
          KZZ   v6   CP   v7

END
```

The v's stand for the real numbers to be substituted for the appropriate property value. Properties which are not required for the analysis need not be entered. For example, the allowable stresses (ST, SC, SS) need not be defined if optimization and/or margin of safety calculations are not performed. Table 2 defines the keyword values for the material cards.

TABLE 2

MATERIAL KEYWORD DEFINITIONS

E	-	Young's Modulus	A11-A33	-	Laminate Moduli
G	-	Shear Modulus	A1		
U	-	Poisson's Ratio	A2		Thermal Expansion
RHO	-	Density	A3		Coefficients
A	-	Thermal Expansion Coef	TREF	-	Reference Temperature
ST	-	Allowable Tension Stress	GE	-	Damping Coefficient
SC	-	Allowable Compression Stress	K	-	Thermal Conductivity
SS	-	Allowable Shear Stress	CP	-	Thermal Capacity/Unit Volume
KXX, KXY, KXZ	-	Thermal Conductivities	KYY, KYZ, KZZ	-	Thermal Conductivities

The required material table must be identified for the model being developed. Typically, table identification numbers would be 1 through n since they are used only during the material definition process. The table ID's do not become the equivalent NASTRAN MAT card numbers since CADS will reorder the numbers on output of the bulk data deck.

Following are the property definition commands; they are also shown in Table 3.

```
PID  id  T    v1   NSM  v2

PR2  id  A    v1   J    v2   C    v3   NSM  v4

PB2  id  A    v1   I1   v2   I2   v3   J    v4   NSM  v5   C1   v6   C2   v7
         D1   v8   D2   v9   E1   v10  E2   v11  F1   v12  F2   v13
         K1   v14  K2   v15  I12  v16

PB2A id  OD   v1   T    v2   NSM  v3   P    v4   C1   v5   C2   v6
         D1   v7   D2   v8   E1   v9   E2   v10  F1   v11  F2   v12

PTM6 id  T1   v1   T3   v2   T5   v3   NSM  v4

PQM8 id  T1   v1   T3   v2   T5   v3   T7   v4   NSM  v5

PRTQ id  PHI  v1

PAS  id  K    v1   GE   v2   S    v3

PS82 id  CID  v1   NIP  v2   AR   v3   ALFA v4   BETA v5

PTQ1 id  T1   v1   MID2 v2   I    v3   MID3 v4   T3   v5   NSM v6   Z1 v7
         Z2   v8

PSHE id  T    v1   MID2 v2   I2IT v3   MID3 v4   TST  v5   NSM v6   Z1 v7
         Z2   v8   MID4 v9
```

TABLE 3

PROPERTY KEYWORD DEFINITIONS

TYPE	KEYWORDS	DESCRIPTION
PID	T	thickness
	NSM	nonstructural mass/area
PR2	A	rod cross-sectional area
	J	torsional constant
	C	torsional stress coefficient
	NSM	nonstructural mass/length
PB2	A	cross-sectional area
	I1	area moments of inertia
	I2	(I1*I2 ≥ I12*I12)
	J	torsional constant
	NSM	nonstructural mass
	Ci,Di,Ei,Fi	stress recovery coefficients
	K1,K2	area factor for shear
	I12	area moment of inertia
PB2A	OD	outside diameter of cross-section
	T	pipe wall thickness
	NSM	nonstructural mass
	P	internal pipe pressure
	Ci,Di,Ei,Fi	stress recovery coefficients
PTM6	T1, T3, T5	membrane thicknesses at the vertices of the element
	NSM	nonstructural mass
PQM8	T1, T3, T5, T7	membrane thicknesses at the element corners
	NSM	nonstructural mass
PRTQ	PHI	list of azimuthal coordinates for stress recovery (degrees: 14 max)
PAS	K	elastic property value
	GE	damping coefficient
	S	stress coefficient
PS82	CID	coordinate system for material
	NIP	number of integration points
	AR	max aspect ratio

TABLE 3

PROPERTY KEYWORD DEFINITIONS (Continued)

TYPE	KEYWORDS	DESCRIPTION
	ALFA	max angle between face triangles
	BETA	max angle between midside vectors
PTQ1	T1	membrane thickness
	MID2	bending material table number
	I	area moment of inertia/width
	MID3	shear material table number
	T3	transverse shear thickness
	NSM	nonstructural mass
	Z1,Z2	fiber distances for stress computation
PSHE	T	membrane thickness
	MID2	material table for bending
	I2IT	area moment/membrane thickness
	MID3	transverse material table
	TST	transverse shear/membrane thickness
	NSM	nonstructural mass
	Z1,Z2	fiber distances for stress computation
	MID4	coupled bending-membrane material

NOTES: 1. For more information on these size keywords, refer to the NASTRAN property card descriptions since these keywords follow the NASTRAN terminology for the element property definitions.

2. For each of the property table definitions, only those values required or desired for the element need be defined. Any keywords not defined will be blank or zero on output.

Table 4 is the list of elements and their property type commands

TABLE 4

ELEMENT PROPERTY DEFINITIONS

NATURAL ELEMENT TYPE	NASTRAN ELEMENT NAME	NATURAL PROPERTY COMMAND
CROD	CROD	PR2
R2	CONROD	PR2
B2	CBAR	PB2
TM	CTRMEM	PID
QM1	CQDMEM1	PID
TB2	CTRIA2	PID
QB2	CQUAD2	PID
TB1	CTRIA1	PTQ1
QB1	CQUAD1	PTQ1
RQ4	CTRAPAX	PRTQ
RT3	CTRIAAX	PRTQ
QS4	CSHEAR	PID
QT4	CTWIST	PID
B2A	CPIPE1	PB2A
TM6	CTRIM6	PTM6
QM8	------	PQM8
SO4	CTETRA	
SO6	CWEDGE	
SO8	CIHEX1	PS82
SO20	CIHEX2	PS82
AS	CELAS1	PAS
TB3	CTRIA3	PSHE
QB4	CQUAD4	PSHE

The element sizes and materials are applied to previously defined element groups through the use of the tables described above. The five commands used to apply these attributes to the elements are:

```
GROUP     id     PTYPE   id    MID   id
PLIST     start  pid     pid   ---
MLIST     start  mid     mid   ---
CHANGE    PID    id      MID   id
ELEMENTS  e1  TO  e2  BY  e3
```

The GROUP command defines the default sizes and material properties to be applied to the entire group of elements. Any PLIST, MLIST, or CHANGE commands which follow a GROUP command selectively change the sizes or properties of certain elements in the group called out by the GROUP command. Any number of PLIST, MLIST, and CHANGE commands may follow a GROUP command. They are processed for that group until an END or another GROUP command is encountered. The PTYPE parameter refers to the PR2, PB2, PID, etc., keywords. The MID parameter refers to a previously defined MAT1,MAT2, MAT4, or MAT5 material table. The id's are integer group or table identification numbers. For example the commands

```
MAT1     1  E     10.0E6  G    3.5E6
MAT1     2  E     15.0E6  U   0.30
PR2     11  A     0.25
PR2     12  A     0.15
PID      1  T     0.50
GROUP    3  PR2   11      MID  1
CHANGE   PID 12   MID     2    ELEMENTS  20  TO  40  BY  2
MLIST   10  2   2   2   2   2   2
GROUP    1  PID   1       MID  2
```

define two isotropic (MAT1) tables (numbered 1 and 2) with the given properties. In each case, the third property is calculated from the other two using the standard G = E/2(1 + U) formula. Two axial rod size (PR2) and one membrane size (PID) tables are also defined with rod areas of 0.25 and 0.15 and a membrane thickness of 0.50. The first GROUP command calls out the previously generated rod group 3 and sets the default size and material tables to 11 and 1, respectively, for the elements in the group. The CHANGE command changes the size and material tables to 12 and 2 for ELEMENTS 20 through 40 by 2; i.e., the even elements 20, 22. . . . 40. The MLIST and PLIST commands operate in the same way with MLIST for material tables and PLIST for property size tables. The MLIST command starts with element 10 and resets its material table number to table 2. This continues for the sequential elements 11, 12, 13, 14, and 15 for each of the remaining table numbers in the command. It is not required that all of the table numbers be the same, but they do have to be defined. The second GROUP command sets the size and material tables for group 1.

For the MLIST, PLIST, and CHANGE commands, the ELEMENTS are numbered (sequentially) in a given group based upon the order in which they were generated in the ELEMENT generation module. The element numbers are the sequential position numbers only. The easiest way to determine these numbers is to plot the group and request the ID numbers. A two-part number will be displayed with the first part as the group number and the second part, the offset or sequential position number.

The END command returns control to the PROPERTY submodule.

4.5.2 ANISOTROPIC PROPERTY SUBPROCESSOR

The function of the ANISOTROPIC subprocessor is to develop the sizes and material property inputs for either orthotropic or anisotropic membrane finite elements using basic lamina characteristics as required input values.

Various groups, of either triangular or quadrilateral membrane or plate finite elements, can have both size and material properties determined automatically. The input requirements, which must be satisfied by the user, are the direction of the zero-degree lamina, the groups of elements which are to be sized, the lamina properties, and the number of plies of a given lamina on the elements of the given group.

The zero-degree lamina direction can be entered by either specifying two node numbers which coincide with the lamina direction or, if no nodes exist in the desired direction, by specifying a vector.

The input and computational order of the module is as follows. First the alignment direction of the zero-degree lamina is defined. The second required input comprises sets of cards with the lamina properties and their alignment with respect to the basic direction. These material cards form tables of composite lamina properties. Typical information on each material card is the alignment angle with respect to the zero-degree laminate, the longitudinal, transverse, and shear modulus, the longitudinal-transverse Poisson's ratio, and the thickness of an individual ply along with thermal, moisture, and allowables data. Finally, the PLIES card defines which elements of the affected group contain a given number of plies.

The ANISOTROPIC subprocessor currently operates with the following restriction: the basic plane of the elements to be sized must be the X-Y plane.

The ANISOTROPIC subprocessor commands are:

```
BASIS   NODE    n1  n2 BASIS   VECTOR  x1   y1   x2   y2
CID     id      EL=v1    ET=v2    UL=v3    GL=v4
                AL=v5    AT=v6    BL=v7    BT=v8
                DEN=v9   T=v10    FTL=v11  FTT=v12
                FCL=v13  FCT=v14  FLT=v15  TE=v16  M=v17
GROUP number
PLIES  number  CID=id  LA=angle  MIN=number  MAX=number  ELEMENTS  list
END
```

The BASIS command is used to define the zero-degree direction for the composite lamina. The angle between this basis and the individual element axes will form the material orientation angle associated with each individual element. The basis direction can be defined by two NODES with the zero direction from node one to node two, or it can be defined by a VECTOR. The VECTOR defines the x,y coordinates for the basis direction from point one to point two.

The CID command identifies a table of lamina material properties for the definition of composite laminates using the PLIES command. The basic format of the CID command is CID followed by an identification number and then keywords and values to specify the appropriate material properties. Up to 10 lamina materials may be defined using CID numbers 1 through 10. All keywords are optional and their use will depend on the analysis program being used and its requirements. Typically the EL, ET, UL, GL, and T keywords will be supplied to provide the basic lamina properties. Table 5 lists all of the valid keywords and their descriptions.

TABLE 5

COMPOSITE LAMINA PROPERTY KEYWORDS

KEYWORD	DESCRIPTION
EL	Longitudinal Modulus
ET	Transverse Modulus
UL	Poisson's Ratio - LT
GL	Shear Modulus - LT
AL	Thermal Expansion Coefficient - Longitudinal
AT	Thermal Expansion Coefficient - Transverse
BL	Moisture Expansion Coefficient - Longitudinal
BT	Moisture Expansion Coefficient - Transverse
DEN	Density
T	Lamina Thickness
FTL	Allowable Stress - Tension, Longitudinal
FTT	Allowable Stress - Tension, Transverse
FCL	Allowable Stress - Compression, Longitudinal
FCT	Allowable Stress - Compression, Transverse
FLT	Allowable Stress - Shear (LT)
TE	Reference Temperature
M	Moisture Content (Percent)

The GROUP command defines the group of elements against which the PLIES commands will be processed. Basically, sets of GROUP and PLIES commands are used to define the lamina associated with each element in the respective groups. The number following the GROUP command is the integer group number for the appropriate elements.

The PLIES command is used to define the number and orientation of the individual lamina which make up the element laminates. The number following PLIES is the number of lamina which will be used for this PLIES command. The CID keyword refers to a previously defined composite lamina property table identification number. That table contains material property data for the lamina. The LA keyword defines the orientation angle for these plies of the given material (CID number) for the specified elements. Often LA will be 0, 45, -45, or 90 for standard 0/±45/90 laminates; however, any real value between 0.0 and 180.0 is valid. Note that the ply-orientation angle is positive clockwise from the basis direction vector. The MIN and MAX keywords are optional and define the minimum and maximum number of plies of the given orientation and properties to be allowed on the specified element list. These would be integer values and would typically be used for optimization programs which vary the number of plies of a given orientation during the optimization process. The ELEMENTS keyword is used to specify the list of elements to which this PLIES command applies. If ELEMENTS is not used, then all of the rest of the elements in the group will be defined with the specified plies. For example, if group 5 has 100 elements in it, the following commands will specify 10 plies of composite material 1 on elements 1 through 50, 25 plies of material 1 on elements 51 through 90, 15 plies of material 1 on elements 91 through 100, and 5 plies of material 2 on elements 1 through 100.

```
CID    1  EL 20.0E6   ET 5.0E6   GL=10.0E6 UL=0.3 T=0.00525
CID    2  EL=10.0E6   ET 10.0E6 GL=5.0E6   UL=0.3 T=0.0115
GROUP   5
PLIES 10   CID 1   LA 45.0   ELEM  1 TO  50
PLIES 25   CID 1   LA 45.0   ELEM 51 TO  90
PLIES 15   CID 1   LA 45.0   ELEM 91 TO 100
PLIES  5   CID 2   LA 90.0   ELEM  1 TO 100
```

The END command terminates the ANISOTROPIC subprocessor and returns control to the PROPERTY submodule.

5.0 OUTPUT MODULE

The OUTPUT module allows the user to output the model in a specific finite element analysis program format. The module will translate information from the geometric data base to a bulk data deck through the use of a separate translator interface for each program. The prompt string is ? OUTPUT and the following commands are valid.

BEGIN - used to start a translator processor

END - ends the OUTPUT module and returns to the Executive Monitor

The BEGIN command format is

BEGIN processor OUTPUT unit

The "processor" keyword is required to define the type of translation to be performed. The valid processor types are:

NASTRAN	-	for NASTRAN Bulk Data Decks
OPTSTAT	-	for OPTSTAT Data Decks
ANALYZE	-	for ANALYZE Data Decks

The OUTPUT keyword defines the Fortran unit to which the program will write the data deck. The default is unit 20; if another unit is used it must be specified as an integer number after the OUTPUT keyword. Typically this keyword is not used and the standard CADS default unit is used. Although the CADS software will do its best to translate the geometry to the requested analysis format, in some cases this may not be appropriate. For example, a NASTRAN data deck may have been read into the geometry file, but it may be output as an ANALYZE model. However, since these two programs do not have a one-to-one correspondence between element types, the resulting ANALYZE model may not be appropriate to the analysis task being undertaken. The user must be careful (when translating between analysis program formats) to ensure that the resulting models are compatible.

The OUTPUT processor will prompt for the file name to be used to save the bulk data output. The prompt is:

ENTER program name OUTPUT FILE NAME NOW OR END TO STOP

where the program name will be NASTRAN, ANALYZE, or OPTSTAT as defined on the BEGIN command and the valid responses are:

end: to end the OUTPUT module and return to the Executive Monitor
name: output bulk data card image file name (max. 40 characters)

A check is made on the file name to see if that file exists and if it does the following confirmation prompt is given:

FILE name
ALREADY EXISTS SHOULD IT BE REUSED (Y/N)?

the valid responses are:

yes: existing file will be written over
no: routine will ask for a new file name

This question is used to help insure that existing data is not accidently destroyed.

Since the CADS program works with the geometry type data for the most part, the finite element analysis program control cards required by NASTRAN, ANALYZE, or OPTSTAT are not completely generated and output by CADS. For NASTRAN the executive and case control decks required to run NASTRAN will be output provided they were read in as part of the NASTRAN input data in the READ module. For ANALYZE and OPTSTAT the control cards are output if they were part of an ANALYZE or OPTSTAT input data deck. In addition, most of the control parameters for ANALYZE and OPTSTAT will be generated from the information contained in the GEOMETRY data base when the new data deck is output by CADS.

6.0 SET MODULE

The SET module is used to define node or element sets for plotting purposes. It is a powerful tool allowing a variety of approaches to define the sets or sections of a model. Three functions are performed in the SET module: general commands, set definitions, and set algebra.

6.1 GENERAL SET COMMANDS

Two types of sets may be defined: node sets (names starting with the letter N) and element sets (names starting with E). A set name is limited to four characters. The prompt string is ? SET. The various general SET commands are:

CLEAR	-	Clears the current set pointer table of node and element set definitions.
PRINT	-	Prints the node or element set members.
LIST	-	Lists the model's nodes and element group tables.
PLOT	-	Sends a set to the DISPLAY module.
HEADER	-	Prints the geometry data file header. (Used for debugging only.)
END	-	Returns to the Executive Monitor

The syntax for each of the above commands is:

```
CLEAR
PRINT   setname
LIST    NODE      input list
LIST    GROUP     input list
PLOT    setname   OFFSET
HEADER
```

The CLEAR command will erase all sets currently defined. It has no keywords.

The PRINT command requires the "setname" of the node or element set to be printed. The command lists all of the members of the given set and is basically used to ensure that the set is composed of the nodes or elements requested. For example, the following command will print (at the terminal) all of the nodes in the node set named N1. The PRINT command lists the actual entities in the given set and is used primarily for debugging purposes. In addition, it can be used to verify that a particular node or element is contained in the given set.

PRINT N1

Once a set is defined, it can be plotted by using the PLOT command under the SET module. The setname is the node or element set to be plotted. Node sets are plotted as individual crosses, while element sets are plotted as lines between node positions and thus are much more understandable. This command begins execution of the DISPLAY module which is described in section 7.0. The keyword, OFFSET, may be added to set up for plotting beam element offsets and additional beam data not typically displayed.

In addition, any of the DISPLAY module commands may be executed directly from the SET module. If this is done, then the last defined node or element set is passed to the DISPLAY module for processing. These commands and keywords are described in section 7.0 (DISPLAY module) of this report. There is one difference between using a DISPLAY module command in SET and using the PLOT "setname" command in SET: PLOT setname specifies any particular previously defined set for plotting, while DISPLAY commands act only on the most recently defined set.

6.2 NODE SET DEFINITIONS

The above paragraphs describe the general SET module commands and functional capabilities. The second function of the SET module is to define node and element sets to specify sections of a model of particular interest to the user. Detailed descriptions of methods for defining node sets are:

```
N-- =   ALL
N-- =   n1, n2 TO n3, n4 TO n5 BY n6
N-- =   SLAB n1, n2, n3 THICK t1 t2
N-- =   SPHERE n1 RADIUS r
N-- =   CYLINDER n1 TO n2 RADIUS r
N-- =   BOX X=x-low, x-high, Y=y-low, y-high Z=z-low, z-high
N-- =   E--
N-- =   operator SUPPRESS TX TY TZ RX RY RZ
```

In the above definitions the n1,---,n6 represent node identification numbers read in from a bulk data deck or generated in the preprocessor. The x, y, and z low and highs are real numbers indicating the ranges on the corresponding axes to be included in the set. The r is a real number for the radius, and the t1 and t2 are real thickness values. Detailed descriptions of these parameters are provided in the following paragraphs.

The ALL keyword will place all of the model's nodes in the given node set. A list of node numbers may also be given, in which case only the nodes contained in the given list will be placed in the node set. The standard list generator, described in section 2.0 (General Information), is used to define the nodes. For example, the command

```
NSET = 1001 1005 TO 1009  1100 TO 1150 BY 2
```

would make up a node set called NSET with nodes 1001, 1005 through 1009, and all even nodes from 1100 through 1150 inclusive.

For the SLAB keywords n1, n2, and n3, are three nodes which define an infinite plane with a local right-handed axis system given by the order of the nodes. The THICK values (t1, t2) are the slab's thickness along the normal to that plane. They are positive, real values which default to 1.0 if not explicitly set by the user. T1 is the distance along the positive normal and t2 is along the negative normal to the plane or slab.

For the SPHERE keyword, n1 is the node which is the center of a sphere with RADIUS r. All nodes inside the given sphere are selected as members of the set.

The CYLINDER keyword defines a cylinder with RADIUS r along a line from n1 to n2. All nodes inside the cylinder are selected for the set.

For the BOX keyword, the low and high values are distances along the given axes for a rectangular box. For example, the command

```
NS1 BOX X  5.0, 15.0   Y -7.5 20.0
```

makes a node set called NS1 from all the nodes in the model which lie within a rectangular box from 5.0 to 15.0 on the x axis, -7.5 to 20.0 on the y axis, and from $-\infty$ to $+\infty$ on the z axis. The default for an unspecified axis is $-\infty$ to $+\infty$.

A node set from a previously generated element set may be defined by naming the desired element set. For example

```
NS2 = ESET
```

would generate a node set named NS2 which contains all of the nodes from a previously generated element set called ESET.

Finally, node sets may be generated using their associated suppressions or single-point constraints to define which nodes will be in the given set. In this case, an OPEN or CLOSE operator is used followed by the keyword SUPPRESS and the list of suppressions which will be operated on. For instance, the command

```
NST1 = OPEN  SUPPRESS  TX  RX  RZ
```

will make up a node set NST1 with all nodes which have at least a translational x (TX), rotational x (RX) or rotational z (RZ) suppression. The OPEN operator requires that a node have at least one of the given suppressions to be included in the set. The CLOSE operator is the opposite of OPEN in that it requires that a node have all of the specified suppressions before inclusion in a given node set. For example,

```
NST2 = CLOSE SUPPRESS TX TY TZ
```

would define a node set NST2 with only those nodes which have the TX, TY, and TZ suppressions.

6.3 ELEMENT SET DEFINITIONS

Element sets are generally used for most display purposes since they contain the connectivities between the nodes for the specific model. The various element set definition techniques are:

```
E---   =   ALL
E---   =   GROUP g1
E---   =   GROUP g1 ELEMENT e1 TO e2 BY inc
E---   =   GROUP g1 TO g2 BY g3 ELEMENT e1 TO e2 BY inc
E---   =   type
E---   =   ID element numbers
E---   =   operator N---
E---   =   operator type N---
```

The element set name is up to four characters long, and the first character is an "E". The 'type' is the assigned element type relative to the program mode selected by the user. If the user wanted a bending beam element in the NASTRAN mode, it would be CBAR; while for the NATURAL mode it would be B2. Note the "operator" can be either OPEN or CLOSE with the default CLOSE.

The ALL keyword will define an element set containing all of the elements in the given model. This command is generally used for smaller models where a display of the entire model is not so complex that little can be seen. Generally, it is more efficient to work with smaller sections of the model and, for this reason, the GROUP-ELEMENT concept is employed.

The GROUP and ELEMENT keywords are most applicable with models built using the generator functions, although they may also be used with existing NASTRAN models. Basically, as elements are read into the data base from the NASTRAN translator or ELEMENT generator, they are split into GROUPS and ELEMENT OFFSETS. The offsets are sequentially ordered lists of the elements in the particular group. For the generator, the GROUPS are user-defined, based upon the element type. For NASTRAN data decks, the NASTRAN read processor automatically separates the various element types into individual groups. For example, all of the CONROD elements become one group as do the other element types in the model.

Thus, element sets may be defined based upon a single group, lists of groups, or combined lists of groups and elements. All of the lists use the standard CADS list generator. For example, the command

EST1 = GROUP 10

would define an element set EST1 composed of the elements in group 10. The command

EST2 = GROUP 10 11 12 TO 14 ELEMENTS 1 TO 10 BY 2

will define set EST2 containing the odd elements from 1 through 10 (i.e., 1, 3, 5, 7, and 9) in groups 10, 11, 12, 13, and 14.

In addition to defining element sets using the GROUP and ELEMENT keywords, element sets may be defined by element type, identification numbers, and from node sets.

An element set, called ESET, containing all of the CQDMEM1 elements in the model, would be formed using the command

ESET = CQDMEM1

In this case the program communication type defined in the initialization process would have to be NASTRAN so that element names would follow the NASTRAN naming convention.

An ID keyword for element sets is used to define sets of elements made up of a list of specific element numbers or identifications. For instance,

ES1 = ID 1001 1002 15001 TO 15105

will make an element set ES1 containing the elements numbered 1001, 1002, and 15001 through 15105.

Finally, element sets may be defined from previously generated node sets using a combination of operator and/or type parameters. The valid operators are OPEN and CLOSE. OPEN requires that at least one of the element's nodes be in the given node set before the element is placed in the element set. CLOSE requires that all of the element's nodes are in the node set before it is added to the element set. CLOSE is the default. For example

ESET CL N1

would make up an element set ESET containing all elements which have all of their nodes in the node set N1. This command also illustrates the typical two character abbreviations used throughout the program as well as the free format input of the program. The input requires just a blank between variables, although commas and equal signs may be used to increase the clarity of the command lines.

In addition to OPEN and CLOSE, an element type can be used to limit a new element set from a node set. For example,

E1 = OPEN CONROD N5

will make an element set E1 containing all CONROD elements which have at least one node in node set N5. Since the CLOSE operator is the default, the following two commands are equivalent:

```
E3 = CQDMEM1 N6
E3 = CLOSE CQDMEM1 N6
```

In each case the element set E3 contains only the CQDMEM1 elements which have all of their nodes in node set N6.

6.4 SET ALGEBRA COMMANDS

The third function of the SET module is to perform set algebra on previously defined sets in order to generate a new set. Sets can be intersected (I), excluded (E), and unioned (U) in the SET module. In using these commands all sets must be of the same type, i.e., either all node or all element sets. Also, set names may be overwritten to limit the number of names being tracked. The algebra commands operate the same way for either the node or element sets. More detailed descriptions are given below.

The format for these commands is:

I - intersection:	NEWSET = OLD1 I OLD2
E - exclusion:	NEWSET = OLD1 E OLD2
U - union:	NEWSET = OLD1 U OLD2 U -- U OLDn

The intersection command defines a new set by the intersection of two old, previously defined sets. For node sets the following example will define two node sets, N1 and N2 (with nodes 1 through 50 and nodes 30 through 70). The intersection of these two sets will form a new set, NST (with nodes 30 through 50).

```
N1     1 TO 50
N2    30 TO 70
NST   N1   I N2
```

Similarly, the exclusion command will exclude the first set from the second set on the command line. An element set example is:

```
E1  =  GROUP  1  ELEMENT  1  TO   20
E2  =  GROUP  1  2  EL    1  TO  100
EST =  E2  E  E1
```

In this case a new set, EST, will be formed by taking or excluding element set E1 from set E2. Set EST will contain group 2 (elements 1 through 100) and group 1 (elements 21 through 100).

Finally, a group of sets can be unioned or combined to form one large set. Up to 49 sets can be unioned in one command although typically no more than 4 to 5 are actually used at once. In the example below, node sets N1, N2, N3, and N4 are combined into one set which is overwriting the old set, N1.

```
N1  =    1   TO   50
N2  =  101   TO  150
N3  =  201   TO  250
N4  =  301   TO  350
N1  =  N1  U  N2  U  N3  U  N4
```

The final set, N1, now contains nodes 1 to 50, 101 to 150, 201 to 250, and 301 to 350. By executing these commands in different combinations, even a very complicated model can be reduced to a manageable and visable display size in a few steps.

7.0 DISPLAY MODULE

The DISPLAY module is used to interactively display the model at the terminal. A variety of commands are available to obtain a particular display or plot format. The function of the display module is to allow the user to readily obtain the displays required to debug and analyze a structural model. This module uses a set of commands to perform its functions. In particular, the PLOT command has a large variety of keywords which provide extensive display options. Details of these commands and keywords are given in the following paragraphs.

7.1 DISPLAY COMMANDS

The DISPLAY module commands are:

```
MARGIN  /  NOMARGIN
TITLE
ROTATE X=value Y=value Z=value SD=value DEFORM SCALE=factor
DISTORT H factor V factor
RETURN
SET
EDIT
DEFINE set
LIST NODE list
LIST GROUP list
CASE number
MODE type
DEFORM/NODEFORM  SCALE=factor
STEP number
ATTRIBUTE
PLOT  keywords
GRAPH
COLOR
END
```

The MARGIN command provides the user with the capability to have square labeled or rectangular nonlabeled displays. The margin nformation consists of rotation angles, distortion factors, sight distance, model reference, date and time, along with what was displayed on the view. Once set it remains set until turned off by the NOMARGIN command. The default is MARGIN.

The TITLE command defines a title line of up to 72 characters which is printed at the top of each plot. Once defined it remains on until a blank title is entered. The title information is entered on the line following the TITLE command.

The ROTATE command defines the amount of rotation about an axis in degrees. Input may be in any order and only values which change need be re-entered. Once set the rotation remains set until changed. The following key-words are available for the ROTATE command.

X:	the 'value' defines the X rotation
Y:	the 'value' defines the Y rotation
Z:	the 'value' defines the Z rotation
SD:	the 'number' is the sight/distance ratio S/D. Default is 1.0E+06.
DEFORM:	plot the deformed model
SCALE:	use the factor to scale the deformations

The rotations are performed about the X, Y, and Z terminal display axes. The Z axis is positive up on the terminal screen; the Y axis is positive to the right on the terminal screen; and the X axis is positive out of the terminal screen toward the user. The initial default is to rotate about the Y and Z axes by 45.0 degrees. All ROTATE command inputs, however, rotate the given number of degrees from X=0.; Y=0.; Z=0.; i.e., the rotations are not cumulative.

The SD sight/distance ratio has the effect of zooming in on the model or moving the user closer to the display. The DEFORM and SCALE keywords multiply the displacements by the SCALE factor so that the DEFORMED model shape will be displayed. They act similar to the DEFORM command which will be described in a subsequent paragraph.

The DISTORT command distorts the horizontal and/or vertical axis of the plot to take full advantage of the display window. Keywords are H and V where the factor defines the scale factor for the horizontal axis and vertical axis, respectively. The factors should be between 0.0 and 1.0. Once set the distortion remains on until reset to H 1.0 V 1.0.

The RETURN command terminates the module's execution and returns to the SET module.

The SET command acts as a RETURN command in that it calls the ? SET module at which time any of the ? SET commands (defined in section 6.0) may be used to define new sets. Once a new set is defined any of the ? DISPLAY commands may be entered. This will return the user to ? DISPLAY with the plotting set now defined as the last node or element set described in ? SET.

EDIT brings in the commands for editing the node and element model data. These commands are described in section 8.0

The DEFINE command is used to define a new plotting set directly from ? DISPLAY. Only one set may be defined, but any of the standard element definitions may be used. For example

DEFINE GROUP 1 TO 5

will define a new plotting set containing the elements in groups 1 through 5, internally named ESET, which can be plotted, rotated or displayed using any of the other ? DISPLAY commands.

The commands

```
DEFINE  CONROD
DEFINE  ID  10111  10121  10300
```

are additional examples of ways to define new sets for plotting purposes. More details can be found in section 6.0 (SET module).

The LIST command acts similar to the LIST command under the SET module in that it lists nodes or element groups at the terminal. Its syntax and function is the same as in the ? SET module.

The CASE command is used to specify the load case identification or mode shape number for analysis output displays. The number is the integer number used to identify the case to the analysis program. For NASTRAN this number is the load subcase or mode shape number defined by the user in the NASTRAN data deck. For ANALYZE and OPTSTAT it is the sequential number of the external load case from their input data decks.

The MODE command in the DISPLAY module is used to define or redefine the type of analysis output to be processed. The MODE command is also valid under ? ATTRIB in the ATTRIBUTE submodule (described in section 7.2). The valid output mode types are STRESS, FORCE, DISPLACE, and EIGEN. These represent the element stresses and forces, the grid-point displacements and the eigenvector data. The format is MODE followed by at least the first four characters of STRESS, FORCE, DISPLACEMENT, or EIGENVECTOR.

The DEFORM/NODEFORM command performs the same function as the DEFORM keyword on the ROTATE command. DEFORM will add the grid point displacements or eigenvector deformations, multiplied by the given scale factor, to the original grid point locations. This deformed shape will be displayed the next time a PLOT command is issued. Once a DEFORM command or keyword is issued, all future displays will be deformed until the NODEFORM command is issued. NODEFORM is a one-word command which turns off the deformation displays. For example, the four commands listed below would plot a deformed shape based upon the displacement mode for load case number 2.

```
CASE   2
MODE  DISPLACE
DEFORM  SCALE  10.0
PLOT
```

Once these commands are issued, another model orientation could be obtained using the following ROTATE command:

ROTATE Y 45.0 Z 60.0

This command would rotate the model 45 degrees about the y axis and 60 degrees about the z axis. Note that these rotations are not cumulative; any new rotation goes from the original orientation and not from the previous model orientation. After this ROTATE command is processed and a new PLOT command is issued, the display will be DEFORMED since that switch is still on and will not go off until NODEFORM is specified.

To plot the deformed on the undeformed shape the PLOT command with the BOTH keyword should be used.

The STEP command is used to specify the time step of the NASTRAN output results to be displayed as contours, values, or deformed shapes. The STEP command is followed by the time step increment number for the particular set of output results needed for the display. For example:

STEP 5

would retrieve the fifth set of dynamic output results for the following displays of stresses, forces, or displacements. Once set the time step value remains in effect until reset by another STEP command.

The ATTRIB command starts the ATTRIBUTE submodule which is used to define the analysis output data components to be displayed at the terminal. It is defined in section 7.2.

The PLOT command causes the execution of a plot request to be displayed on the screen. Any combination of information can be displayed. The keywords are in effect only for the given display and are described in section 7.3. When multiple keywords are specified for a given plot, the amount of information can rapidly overwhelm a typical display terminal. In those cases, multiple displays should be made to limit the amount of data on any one plot.

The GRAPH command in the DISPLAY module is used to plot values as an x-y graph. Basically, this command plots up to five different sets of values in the form of an x-y graph. The user can provide title and legend information which is included with the display. It functions like the ATTRIBUTE command in that it initiates the GRAPH submodule in the same way ATTRIB initiates the ATTRIBUTE submodule. It is described in section 7.5.

The COLOR command turns on the color processing for plane elements. It will fill in each different type of element with a different color. It is most effective with the BREAK option of the PLOT command for displaying each element separated from its neighbors. It is a time-consuming command, especially on non-raster terminals and thus should be used with discretion.

END terminates the module's execution and returns to the Executive Monitor.

7.2 ATTRIBUTE SUBMODULE

The ATTRIBUTE submodule controls the specification of output analysis components for display. Its prompt string is ? ATTRIB and the valid commands are:

```
PROG  name
MODE  type
CLEAR name  ALL
HELP  name
NAME  components
END
```

7.2.1 ATTRIBUTE COMMANDS

The PROG command in the ATTRIBUTE submodule is used to specify the type of analysis output data to be retrieved from the POST data base for display. The valid type names are NASTRAN, ANALYZE, or OPTSTAT. The requested name must match the type of data stored on the attached POST data base or else an error message will be displayed.

The MODE command specifies the type of output data to be displayed. The valid types are STRESS, FORCE, DISPLACE, and EIGEN. These correspond to the element stress or force data or the grid displacement data or eigenvector data from the analysis program. The following example would specify the stress data type.

```
MODE  STRESS
```

The CLEAR command will erase or clear the internal switches for either the specified name or for all previously defined components. The valid names are the element type names for the given communication mode and the word NODE for any grid point data types. For example, the following commands will clear the CONROD, CQDMEM1, and CSHEAR component definitions:

```
CLEAR  CONROD
CLEAR  CQDMEM1
CLEAR  CSHEAR
```

If all the previous definitions should be cleared use

```
CLEAR  ALL
```

The HELP command in this submodule should be followed by an element name. For an ANALYZE type 3 element use:

```
HELP 3
```

This command will then print, at the terminal, the valid components for that element for the particular mode. For instance, the following commands define the stress mode and request HELP for the ANALYZE element type 3:

```
PROGRAM ANALYZE
MODE  STRESS
HELP  3
```

The HELP command would return the following data:

```
THE  STRESS  COMPONENTS  FOR  THE  3  ELEMENT  ARE:
SX  SY  SXY  EFS1  ENER  MS
```

This command summarizes the information presented in Tables 6-8 which have all of the valid data components for each element for each program type.

TABLE 6

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE or EIGEN	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
		RX	Rotation in the X direction
		RY	Rotation in the Y direction
		RZ	Rotation in the Z direction
FORCE	CROD	F	Axial force
		T	Torque
	CONROD	see CROD	
	CBAR	MA1	Bending moment at A1
		MA2	Bending moment at A2
		MB1	Bending moment at B1
		MB2	Bending moment at B2
		SHR1	Shear at point 1
		SHR2	Shear at point 2
		F	Axial force
		T	Torque
	CTRMEM	none	
	CQDMEM1	none	
	CTRIA2	MX	X - moment
		MY	Y - moment
		TM	Twist moment
		SHRX	X - shear
		SHRY	Y - shear
	CQUAD2	see CTRIA2	
	CTRIA1	see CTRIA2	
	CQUAD1	see CTRIA2	
	CTRAPAX	R1	Radial force at ring 1
		R2	Radial force at ring 2
		R3	Radial force at ring 3
		R4	Radial force at ring 4
		T1	Tangential force at ring 1
		T2	Tangential force at ring 2
		T3	Tangential force at ring 3
		T4	Tangential force at ring 4
		Z1	Axial force at ring 1
		Z2	Axial force at ring 2
		Z3	Axial force at ring 3
		Z4	Axial force at ring 4

TABLE 6 (CONTINUED)

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
	CTRIAAX	See CTRAPAX	
	CSHEAR	F41	Force from point 4 to 1
		F21	Force from point 2 to 1
		F12	Force from point 1 to 2
		F32	Force from point 3 to 2
		F23	Force from point 2 to 3
		F43	Force from point 4 to 3
		F34	Force from point 3 to 4
		F14	Force from point 1 to 4
		K1	Kick force at point 1
		K2	Kick force at point 2
		K3	Kick force at point 3
		K4	Kick force at point 4
		V12	Shear from point 1 to 2
		V23	Shear from point 2 to 3
		V34	Shear from point 3 to 4
		V41	Shear from point 4 to 1
	CTWIST	see CSHEAR	
	CPIPE1	see CBAR	
	CTRIM6	none	
	QM8	none	
	CTETRA	none	
	CWEDGE	none	
	CIHEX1	none	
	CIHEX2	none	
	CELAS1	F	Axial force
	CTRIA3	FX	Force in x direction
		FY	Force in y direction
		FXY	Force in xy
		MX	Moment about x
		MY	Moment about y
		MXY	Moment about xy
		VX	Shear in x
		VY	Shear in y
	CQUAD4	see CTRIA3	

TABLE 6 (Continued)

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
STRESS	CROD	A	Axial stress
		ASM	Axial margin of safety
		T	Torsional stress
		TSM	Torsional margin of safety
	CONROD	see CROD	
	CBAR	SA1	Stress at A1
		SA2	Stress at A2
		SA3	Stress at A3
		SA4	Stress at A4
		A	Axial stress
		SAMX	Maximum stress at A
		SAMN	Minimum stress at A
		SMT	Margin of safety in tension
		SB1	Stress at B1
		SB2	Stress at B2
		SB3	Stress at B3
		SB4	Stress at B4
		SBMX	Maximum stress at B
		SBMN	Minimum stress at B
		SMB	Margin of safety in compression
	CTRMEM	SX	Stress in the X direction
		SY	Stress in the Y direction
		SXY	Stress in the XY direction
		ANG	Principal stress angle
		MAJP	Major principal stress
		MINP	Minor principal stress
		MAXS	Maximum shear stress
	CQDMEM1	See CTRMEM	
	CTRIA2	FDB	Fiber distance bottom side
		SXB	X-stress bottom side
		SYB	Y-stress bottom side
		SXYB	XY-stress bottom side
		ANGB	Principal stress angle bottom
		SMJB	Major principal stress bottom
		SMNB	Minor principal stress bottom
		SMXB	Maximum shear stress bottom
		FDT	Fiber distance top side
		SXT	X-stress top side
		SYT	Y-stress top side
		SXYT	XY-stress top side
		ANGT	Principal stress angle top
		SMJT	Major principal stress top
		SMNT	Minor principal stress top
		SMXT	Maximum shear stress top

TABLE 6 (Continued)

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
	CQUAD2	See CTRIA2	
	CTRIA1	FD1	Fiber distance one side
		SX1	X-stress one side
		SY1	Y-stress one side
		SXY1	XY-stress one side
		ANG1	Principal stress angle one side
		SMJ1	Major principal stress one side
		SMN1	Minor principal stress one side
		SMX1	Maximum shear stress one side
		FD2	Fiber distance two side
		SX2	X-stress two side
		SY2	Y-stress two side
		SXY2	XY-stress two side
		ANG2	Principal stress angle two side
		SMJ2	Major principal stress two side
		SMN2	Minor principal stress two side
		SMX2	Maximum shear stress two side
	CQUAD1	See CTRIA1	
	CTRAPAX	R	Radial stress for the element
		Z	Axial stress for the element
		T	Tangential stress for the element
		ZR	Shear stress for the element
		RT	Shear stress for the element
		ZT	Shear stress for the element
	CTRIAAX	See CTRAPAX	
	CSHEAR	MAXS	Maximum shear stress
		AVRS	Average shear stress
		SM	Margin of safety
	CTWIST	see CSHEAR	
	CPIPE1	see CBAR	
	CTRIM6	none	
	QM8	none	
	CTETRA	none	
	CWEDGE	none	

TABLE 6 (Concluded)

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
	CIHEX1	SX	Normal X stress
		SXY	Shear XY stress
		SMX	First principal
		SMX1	First principal X cosine
		SMX2	Second principal X cosine
		SMX3	Third principal X cosine
		MS	Mean stress
		OSS	Octahedral shear stress
		SY	Normal Y
		SYZ	Shear YZ
		SMY	Second Principal
		SMY1	First principal Y cosine
		SMY2	Second principal Y cosine
		SMY3	Third principal Y cosine
		SZ	Normal Z
		SYX	Shear ZX
		SMZ	Third principal
		SMZ1	First principal Z cosine
		SMZ2	Second principal Z cosine
		SMZ3	Third principal Z cosine
	CIHEX2	see CIHEX1	
	CELAS1	S	Axial stress
	CTRIA3	see CTRIA1	
	CQUAD4	see CTRIA1	

TABLE 7

VALID COMPONENT TYPES FOR ANALYZE DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
STRESS	2	SX	X local element stress
		ENER	Total strain energy in the elem.
		MS	Margin of safety
	3	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		ENER	Total strain energy in the elem.
		MS	Margin of safety
	4	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		EFS2	Effective stress ratio - 2
		EFS3	Effective stress ratio - 3
		EFS4	Effective stress ratio - 4
		ENER	Total strain energy in the elem.
		MS	Margin of safety
	5	SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		EFS2	Effective stress ratio - 2
		EFS3	Effective stress ratio - 3
		EFS4	Effective stress ratio - 4
		ENER	Total strain energy in the elem.
		MS	Margin of safety

Note: These are the only valid element types for ANALYZE.

TABLE 8

VALID COMPONET TYPES FOR OPTSTAT DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
STRESS	2	SX	X local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ENER	Total strain energy in the elem.
		OPTT	Optimized area value
	3	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of trans. tension to ALS1
		ALS4	Ratio of trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the element
		OPTT	Optimized thickness value
	4	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of Trans. tension to ALS1
		ALS4	Ratio of Trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the element
		OPTT	Optimized thickness value
	5	SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of trans. tension to ALS1
		ALS4	Ratio of trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the element
		OPTT	Optimized thickness value

TABLE 8 (Concluded)

VALID COMPONET TYPES FOR OPTSTAT DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
	3	LAM	Total number of layers
		THK	Total thickness in 0° direction
		AEX	Proportion of fibers in 0° direction
		THK9	Total thickness in 90° direction
		AEY	Proportion of fibers in 90° direction
	4	LAM	Total number of layers
		THK	Total thickness in 0° direction
		AEX	Proportion of fibers in 0° direction
		THK9	Total thickness in 90° direction
		AEY	Proportion of fibers in 90° direction

NOTE: These are the only valid element types for OPTSTAT. The second set of components for the type 3 and 4 elements is output when layered composite elements are used. These components replace the X, Y, and XY local element stress components.

Once the PROGRAM MODE has been defined, data components (which are to be displayed) can be specified. The user enters the appropriate element name followed by a list of component names. Examples of different component definition commands for various mode types are:

```
PROG NASTRAN
MODE  STRESS
CONROD  A
CQDMEM1  SX  SY  SXY
CTRMEM   SX  SY  SXY

PROG NASTRAN
MODE  FORCE
CBAR    MA1  MA2
CQUAD1  MX   MY

PROG NASTRAN
MODE  DISPLACE
NODE  TX  RX  RY  RZ

PROG  ANALYZE
MODE  STRESS
3     EFS1  ENER  MS
```

For the stress mode, the axial stress on the CONROD and the plane X, Y, and XY stresses on the CQDMEM1 and CTRMEM elements would be defined. For the element force mode, the bending moments at A1 and A2 for the CBAR elements and the X and Y moments for the CQUAD1 elements are defined. For the displacements the translational X and rotational X, Y, and Z components are defined for value displays in the DISPLAY module. Finally, for the ANALYZE example, the stress components EFS1, ENER, and MS are defined for the ANALYZE triangular membrane element number 3. Note that in any one use of the ATTRIBUTE submodule, only one MODE type may be defined.

The END command is used to signal completion of the ATTRIBUTE submodule for return to the DISPLAY module.

7.3 PLOT COMMAND

The PLOT command in the DISPLAY module performs the actual plotting to the terminal screen. The keywords available on the PLOT command provide labelling, shrink, axis, hidden line and a variety of other display options. These keywords apply only to the current display and thus must be entered each time information is requested. Some keywords are mutually exclusive and these are noted in the keyword descriptions.

Valid PLOT command keywords are listed and described as follows:

DASH NODE ELEMENT ID AXIS LABEL LM=case LF=case
COORD TYPE=name BREAK SHRINK=value SCALE=value
WINDOW=number PORT=number CONTOUR=name LEVELS=number
SIZE=parameters MATERIAL=parameters SUPPRESS=parameters
HIDE BOTH STRESS FORCE DISPLACE EIGEN OV=parameters

DASH: Displays the membrane elements as dashed instead of solid lines.

NODE: Displays the node numbers.

ELEMENT: Displays the element numbers.

ID: Displays the element numbers as a Group-Offset number as opposed to the standard element number. For example, a NASTRAN element (numbered 20010) read as a Group-Offset would be numbered Group 2 Offset 10. This would be shown as 2-10 on the element.

AXIS: Plots the element local axis on the displayed planer elements.

LABEL: Displays the element type label (CQDMEM1, CONROD, etc.,) on the plotted element.

LM: Plots the applied moments as values at the nodes for the case specified by the case parameter following LM.

LF: Plots the applied forces as values at the nodes for the case specified by the case parameter following LF.

COORDINATE: Displays the x, y, z, coordinates of the nodes. Abbreviated COOR.

TYPE: An element type name is given which restricts output values displayed to that element type. For example, if TYPE CONROD is specified and element numbers are requested, only the CONROD numbers will be displayed. However, the other element types in the display set are still plotted, but their element numbers are not shown.

BREAK: Displays the elements shrunken about their centroids, disconnected from their nodes.

SHRINK: Used to specify the shrunken element size. The factor is between 0.0 and 1.0 and is the value by which the element is scaled. The default is 0.8 for an element, i.e., 80 percent of the original size. SHRINK must be used along with BREAK.

SCALE: SCALE is a multiplication factor used to scale the output element stress and force displays. The default value is 1.0. Any value may be entered.

WINDOW: Windows a section of the model about a point digitized in the previous plot, as described in section 7.4. An integer number follows the keyword to specify the window to be displayed.

PORT: An integer value follows the keyword to define the port box to be enlarged from the port definition in a previous plot. Further information follows these keyword definitions in section 7.4.

CONTOUR: Contours data for surface contours. Valid parameters are listed below. The parameters are:

STRESS
FORCE
MATERIAL

For the material parameter, any of the composite laminate properties may be specified. For contour plots the STRESS and FORCE contours will display the first component type defined for each of the planer element types. For example, if the following ATTRIB commands were used to call out stresses on the CQDMEM1 and CTRMEM elements, the STRESS contours would be of the SY element stresses.

```
MODE STRESS
CQDMEM1  SY
CTRMEM   SY
END
PLOT  CONTOUR  STRESS
```

The user should be careful in defining component values for contouring to ensure that similar components are processed. Generally, this means that element x-stresses would go together on one contour display while the y-stresses would be on another.

For the MATERIAL parameter a second variable is entered after MATERIAL to specify the material value to be processed. The valid types are the MAT2 laminate properties A11 through A33. For example, the command

PLOT CONTOUR MATR A11

would contour the A11 material terms while

PLOT CONT MATR A23

would do the A23 terms for a composite cover.

LEVELS: The LEVELS parameter is followed by an integer number which is the number of contour levels to be used on a contour plot. Any number between 1 and 15 may be used. When the parameter is used, the program then prompts for a list of LEVEL values. These values are real numbers which will form the actual contour levels for a contour display. By default the CADS software will establish between 10 and 15 levels at even spacings using factors of 1.0, 2.5, or 5.0.

SIZE: Currently displays the major size value of the elements. For example, rod areas, plate thicknesses, and other similar values are displayed. The SIZE, MATERIAL, and OV keywords are mutually exclusive since they overlap their data values on a display. The valid SIZE parameters are listed below. (Any number of values may be entered.)

T: Primary element thickness value for any element.
I: Area moment of inertia/unit width (QUAD1)
T3: Transverse shear thickness (QUAD1)
I1: Area moment of inertia (BEAM)
I2: Area moment of inertia (BEAM)
J: Torsional constant (BEAM)
A: Axial rod or beam element cross-sectional area

MATERIAL: Used to define element material table components to be displayed. Any number of the following parameters may be requested for display on any one plot. The MATERIAL, SIZE, and OV keywords are mutually exclusive.

E:	Isotropic elastic modulus
G:	Isotropic shear modulus
U:	Isotropic Poisson's ratio
A11, A12, A13, A22, A23	NASTRAN MAT2 composite laminate properties
A33	
E1, U1, E2, U2	Elastic modulus and Poisson's ratio for the transverse (MID1) and bending (MID2) properties of the NASTRAN CQUAD1 element

SUPPRESS: The SUPPRESS keyword is followed by a list of parameters used to identify the nodes to be highlighted based upon their respective degree-of-freedom suppressions. The valid parameters are:

TX, TY, TZ	Translation X, Y, Z
RX, RY, RZ	Rotational X, Y, Z
TA	Translational arrows to be displayed
RA	Rotational arrows to be displayed.

Suppress prints the node numbers which have the corresponding freedoms suppressed in the model.

For instance, if SUPPRESS TX TY were specified, then all nodes with only TX and TY suppressed would be numbered on the display. The TA and RA parameters will display a small set of axis arrows on each node in the direction of the freedom which is suppressed. For example, if a node is fixed in Y and Z, a small arrow will be placed at the node in the Y and Z directions to indicate this suppression.

OV: The CBAR OFFSET VALUE (OV) keyword is followed by a list of parameters to be displayed for the CBAR offset values. These parameters are:

PA: Pin flag at point A

PB: Pin flag at point B

Z1A, Z2A, Z3A: Components of stress offset vector at point A

Z1B, Z2B, Z3B: Component of stress offset vector at point B

HIDE: Requests the display as a hidden line plot making use of the NASA/DRYDEN general hidden line package. For display, the hidden line processor is limited to about 4000 elements at a time, depending on the complexity of the element.

BOTH: The BOTH parameter may be used only after a DEFORM command has been processed. BOTH will plot the DEFORMED shape on top of the undeformed shape for a given deformation set. The deformed shape is in dashed lines and the undeformed shape is in solid lines.

STRESS: The STRESS parameter, without CONTOUR before it, will display the element stress values on the individual elements. The components requested in the ATTRIBUTE submodule are displayed for the given load case. The component labels are shown in the margin for each element type requested.

FORCE: Displays the element FORCE values on the element, similar to the STRESS parameter.

DISPLACE: Similar to the STRESS parameter, except DISPLACE will display the grid point displacements at the nodes.

EIGEN: Displays the EIGENVECTOR mode shape data at the nodes.

7.4 PLOT END PROCESSING

After a plot is completed, one of the following seven characters may be entered to continue processing. Each character performs a different function as outlined in the following paragraphs.

A b (blank) character will erase the current display and return control to the ? DISPLAY level for another plot.

A W character will initiate the window definition process for blowing-up displays. After the W is entered, the cross hairs will appear on the terminal. The cross hairs are then positioned to the center of the area to be blown up and an integer character (1-9) is entered. This action defines the area for the window numbered by the specific character, i.e., 1. The cross hairs can be moved to another section of the display so that another window can be defined, i.e., 2. This process can be carried out until either nine windows are defined or the letter R is input. The R erases the screen and returns control to ? DISPLAY. At this point any window may be displayed using the PLOT WINDOW n command. A window defines 10 percent of the original plot centered about the cross hairs. As each window is defined, a 24-character title may be input. The title is entered along the bott m of the screen and is displayed in the margin of the window plot.

A P character will initiate window definitions through a port or box function. Again, the cross hairs will appear and should be positioned at a lower left corner of a box to be blown up. An integer character is entered and the cross hairs are then positioned to the upper right corner of the box to be blown up and a character is entered. These two points define a box numbered by the given integer value, which can be expanded as a window. Up to nine boxes may be defined as with the window (W) processor. The R (return) character is used to go back to the ? DISPLAY level. The port is then displayed as PLOT PORT n as for the window processor. Again, a title is entered for each port as for each window.

The K character keeps the current display and lets the user add more data to the display. After the K is entered, the cross hairs appear. Three character inputs are allowed: N, E and R, where N will add the node numbers, and E will add the element numbers to the current display. The R executes the N and/or E commands and returns control to the end of the plot so that a blank or other valid character can be entered. The node or element numbers are then displayed on the current plot for all of the elements or nodes. This command is the same as using the NODE or ELEMENT keywords on a PLOT command.

The V character, for VIEW, will allow the expansion of a boxed area of the current display to another area of the same display without erasing the screen. After the V is entered the cross hairs appear. They are moved to the bottom left corner of a box area to be expanded, a D is input, and the cursor is moved to the upper right corner of the box and another character is entered. This defines the first box. Another box is defined in a clear section of the display in the same way and then an R is input for RUN. The first box is then distorted into the second box, and the cursor returns to the home position so that another blank, W,P,V, or K can be entered. If any character besides a D is used to define the box, a square box is plotted into a square box based upon the side defined by the crosshair inputs.

For WINDOWS, PORTS, and VIEWS, boxes are drawn on the original display outlining the area to be redrawn by the command.

For composite layered elements, that is, those elements for which multiple material layers are defined, an L character may be entered. The L character will request detailed composite layer information displays. Once L is entered the cross hairs are displayed and the user positions them on a composite element. An integer character is entered to number the element selected. Up to 9 different elements may be selected at one time. After selecting the elements to be displayed, an R is entered and a detailed composite layer display is shown.

If errors are discovered in the element or grid point information through the displays, they may be corrected by entering the EDIT module. The EDIT commands are activated by issuing the EDIT command after a display is completed. The EDIT module commands are described in section 8.0.

7.5 GRAPH PROCESSING

The GRAPH submodule of the DISPLAY module is used to display sets of values as an X-Y graph. The user either specifies up to five sets of values, titles, and legends which are used to build the actual X-Y display or enters lists of real values from the terminal for the X-Y display. The prompt string for the GRAPH submodule is ? GRAPH.

The GRAPH submodule commands are

```
TITLE
XTITLE
YTITLE
CURVES    number
CASE      list
STEP      list
XVALUE    number    AXIS  x,y,z       NODES   list
                    TIME                       all
                                      SET     name
YVALUE    number    DISPLACE  tx, ty, tz, rx, ry, rz
                    EIGEN     tx, ty, tz, rx, ry, rz
LEGEND
EXECUTE
TERMINAL
END
```

With the exception of the TERMINAL command all of the other GRAPH commands are used to define the data to be displayed from the CADS data bases. The TERMINAL command is used to define input lists for display from the terminal itself.

The TITLE command is used to enter a title for the next x-y graph. The title will be printed at the top of the graph and may be up to 72-characters long. Once set, this title will remain on until a new TITLE command is executed with a blank line for the title. After the TITLE command is given a line of character data is input.

The XTITLE command specifies the X-axis title and the YTITLE command specifies the Y-axis title. These are one-line titles of up to 48 characters. The XTITLE is typed horizontally along the X-axis, while the YTITLE is typed vertically along the Y-axis. These titles remain on until a new command is specified defining a blank line.

The CURVES command specifies the number of sets of values to be plotted as separate curves. Up to five curves may be defined at one time and, once set, this number remains on until redefined by a new CURVES command. The number which follows the CURVES command is a integer number between 1 and 5.

The CASE command is the same as the CASE command in the DISPLAY module in that it defines a load case to be used for data display. In the GRAPH submodule, the CASE command is followed by a list of case numbers - one for each curve to be plotted. This permits the user to rapidly compare the results of different analysis load cases. If fewer cases are specified than are called for by the CURVES command, the last case number in the CASE list is used for the remaining curves. For example, the following two commands call for four curves which will use cases 1, 3, 5, and 5 for curves 1 through 4, respectively,

```
CURVES  4
CASE    1  3  5
```

The STEP command is used to list the time step numbers to be displayed as time values for dynamic analysis results. A list of time steps is given. For instance the command

```
STEP  5  TO  20
```

would request that the time step values for steps 5, 6, 7, ... 20 be used as the X or Y values depending upon which parameters were used for the XVALUE or YVALUE commands.

The XVALUE and YVALUE commands specify the sets of values to be plotted as X-Y curves. One set of X and Y values must be defined for each curve. The curve number is given as the number immediately after the XVALUE and YVALUE commands. A maximum of 500 points may be shown for any one curve. For both the XVALUE and YVALUE commands the same set of keywords are used. The AXIS, TIME, DISPLACE, and EIGEN keywords are used to define the type of value to be plotted along the given axis. For example, DISPLACE will plot a displacement component along the axis while AXIS will plot a node coordinate value along the X or Y axis. EIGEN will plot a mode shape component and TIME will plot time step intervals for the X or Y values. Just one of these four keywords may be defined on any one XVALUE or YVALUE command. Following these four keywords are valid parameters which define the components to be used. For AXIS the parameters are X, Y, or Z for those coordinate values. For DISPLACE and EIGEN the parameters are TX, TY, TZ, RX, RY, or RZ which define the translational or rotational degree-of-freedom component to be used. The TIME keyword has no parameters. The NODES or SET keyword is required on all XVALUE and YVALUE commands since they define the nodes to be used in the given X-Y plot. This keyword is followed by a list of nodes, ALL, or a node set name. The list can be a standard CADS list including the TO and BY keywords. ALL is for all nodes, and a node set name will use a previously defined node set for the required graph.

The LEGEND command turns on the legend for the plot. NO LEGEND turns the legend off. The default is on.

The EXECUTE command ends the curve definition process and begins plotting the actual X-Y graph.

The TERMINAL command is used to start the interactive input of values for X-Y plotting from the terminal. CADS will begin by prompting for the X-axis, Y-axis and plot titles, the number of sets of values (curves), and the curve values. The following CADS prompts will request information for the TERMINAL command:

```
HOW MANY CURVES WILL BE GRAPHED (INTEGER NUMBER)?
number

WHAT IS THE GRAPH TITLE (1 LINE, 72 CHARACTERS)?
line one

WHAT IS THE X-AXIS TITLE (1 LINE, 48 CHARACTERS)?
X-title

WHAT IS THE Y-AXIS TITLE (1 LINE, 48 CHARACTERS EACH)?
Y-title

ENTER THE X-VALUES FOR CURVE n (FREE FORMAT, REAL NUMBERS)?
X-values (maximum of 100)
```

ENTER THE Y-VALUES FOR CURVE n (FREE FORMAT, REAL NUMBERS)?
Y-values (maximum of 100)

The last two prompt lines will repeat for curves 1 through n where n is the number given in the first line. Since the CADS free read routine is used to read in the X and Y values there is a limit of 100 values for each array.

After inputting the given values, CADS will display the requested X-Y graph. Once the graph is completed, a blank character is entered and the screen is erased with control returned to ? GRAPH. If additional curves are needed, the TERMINAL command is re-entered and the prompting begins again.

Examples using the X-Y plotter are included in section 12.0, (Sample Sessions).

The END command terminates the GRAPH submodule and return control to the DISPLAY module.

8.0 EDIT MODULE

8.1 EDIT INITIATION

By entering the EDIT command under the Executive Monitor or DISPLAY module the user initiates the CADS functions which allow modification of the current model data. The capabilities of the EDIT module provide for changing, adding, deleting, or listing node, element, size, and material data. In addition HELP and SAVE functions are provided. The prompt string is ? EDIT and the valid commands are:

BEGIN - used to start an edit processor
SAVE - used to save the edited data base
COPY - used to copy to a new file
END - used to end EDIT and return to the Executive Monitor or DISPLAY module

The command formats are:

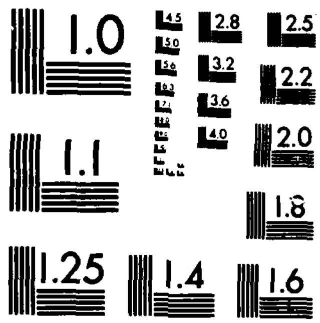
BEGIN processor
SAVE
COPY
END

The "processor" parameter is required since it defines the data type to be changed. The valid processors are:

NODE - for model node data
ELEMENT - for model element data
PROPERTY - for element sizes
MATERIAL - for element material data
CASE - for case control type data

AD-A163 258 CADS - A COMPUTER AIDED DESIGN SYSTEM VOLUME 2 USER'S GUIDE(U) ROCKWELL INTERNATIONAL LOS ANGELES CA NORTH AMERICAN AIRCRAFT.. M C LESS ET AL. AUG 85 2/3

UNCLASSIFIED AFWAL-TR-85-3066-VOL-2 F33615-81-C-3229 F/G 9/2 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

8.2 NODE EDIT PROCESSOR

The NODE processor under the EDIT module provides commands for modifying the current node information. It makes use of the DIRECT node generation capabilities described in section 4.3.1 to add new nodes. The prompt string for the NODE processor is ?EDITND and its syntax is:

```
LIST     node  list
DELETE   node  list
CHANGE   X Y Z S   NODE   list
ADD
END
```

The LIST command will list the current node numbers, coordinates, and suppression information stored on the geometry data base at the terminal. By default all nodes in the data base will be listed. The user may also specify a standard TO/BY list to define the nodes to be listed. For example the command:

```
LIST   1   5   7
```

will list node data for nodes 1, 5, and 7 at the terminal while the even nodes 2, 4, 6, 8, and 10 would be listed if this command was used:

```
LIST   2   TO   10   BY   2
```

The DELETE command requires a node list of at least one node number. This command deletes the specified node(s) from the geometry data base. As for the LIST command a standard TO/BY list or a simple list of numbers can be specified. For example the command:

```
DELETE   1
```

will delete node number 1.

The CHANGE command is used to change the coordinate location or suppression information for the specified nodes. The nodes are specified using a node list following the NODE keyword. The X, Y, and Z parameters are used to define which coordinate values will be changed. Any combination of X, Y, and Z is valid. The S parameter is required for changing the node's suppressions. On the CHANGE command one value must be given for each parameter specified for each node contained in the node list or set.

```
CHANGE   X  Y   NODE  5  6  7
```

will provide for changing the x and y coordinates for nodes 5, 6, and 7. The command:

```
CHANGE   X  Z  S   NODE   1  TO  10  BY  4
```

will provide for changing the x and z coordinates and suppressions for nodes 1, 5, and 9.

After a CHANGE command another line is entered which contains the new node data values for the given list of nodes. Sets of coordinates and suppression values are given for each node. For instance the CHANGE X Y NODE 5 6 7 command would require three sets of two values. These values would be the new X and Y coordinates for nodes 5, 6, and 7. The CHANGE X Z S NODE 1 TO 10 BY 4 command would require three sets of three values. The first two values of each set would be the X and Z coordinates for nodes 1, 5, and 9 while the third value would be an integer defining the suppressions for those nodes. For example the following lines would change the X and Y coordinate for nodes 5, 6, and 7 to (1.5, 2.0), (3.0, 3.0), and (4.5, 4.0) respectively.

```
CHANGE  X  Y   NODE  5  6  7
1.5 2.0   3.0 3.0   4.5 4.0
```

Note that the second line must follow the corresponding CHANGE command and is also input in free format. For the suppressions the constraints are numbered 1,2,3,4,5, and 6 for TX, TY, TZ, RX, RY, and RZ - the translational and rotational degrees of freedom. The numbers corresponding to the required constraints are entered for the nodes. For instance the number 146 would fix the TX, RX, and RZ degrees of freedom for a given node. The following lines would make the X coordinate for node 5 equal to 7.5 and would fix the TX, RX, and RZ degrees of freedom.

```
CHANGE  X  S   NODE  5
7.5     146
```

If a large number of nodes are being changed it may be easier to use the ADD command to change the nodes since the ADD command will write over or replace currently existing nodes if they are respecified. The ADD command calls in the DIRECT node generation routine for defining nodes. All of those commands are valid and they are defined in section 4.3.1.

The END command returns control to the EDIT module.

8.3 ELEMENT EDIT PROCESSOR

The ELEMENT processor under the EDIT module provides commands for modifying the current element connectivity information. The ELEMENT prompt string is ? EDITEL and its syntax is:

```
LIST    GROUP
LIST    GROUP    number  ID     list
LIST    GROUP    number  ELEM   list
DELETE  GROUP    number
DELETE  GROUP    number  ID     list
DELETE  GROUP    number  ELEM   list
CHANGE  GROUP    list    ID     list
CHANGE  GROUP    number  ELEM   list
ADD     GROUP    number
HELP    ELEM
HELP    MAT. type
HELP    PROP. type
HELP    PROP. GROUP = number
HELP    PROP. type  = elem. type
END
```

The LIST command will list element data at the terminal. By default it will list the group numbers and number of elements for each group in the model. All other list commands will list the requested element numbers, types, and connectivities.

The GROUP, ID, and ELEM parameters are used to define a list of element groups and offsets or element numbers to be processed. The ELEM parameter is used to specify a list of the given element numbers, while ID specifies a list of element offsets to be processed for the given group. Some LIST examples follow:

```
LIST   GROUP   1  ID  1 TO  10
LIST   GR      2
LIST   GROUP   5  EL  5001  5002
```

The first command lists element offsets 1 through 10 for group 1. The second command lists all of group 2. The third command lists the elements numbered 5001 and 5002, in group 5. In all three cases the element number, type, and connectivity is listed.

The DELETE command will delete the specified elements from the data base. The GROUP, ID, and ELEM, parameters operate for the DELETE command the same as they do for the LIST command. For example the command:

```
DELETE GR 1    ELEM    1001    1002    1009    1011
```

would delete elements 1001, 1002, 1009, and 1011 of group 1 from the data base. In the same way the element offsets 5, 10, and 15 in group 10 would be deleted using the following command:

```
DELETE    GR    10    ID  5  10  15
```

The CHANGE command will change the element connectivity for the specified element list. The GROUP, ID, and ELEM parameters are allowed. The line after the CHANGE command must contain the new connectivities for the given elements. ɪor example the two lines:

```
CHANGE   GR   1   ID   1  TO  10  BY  2
11 12,  13 14,  15 16,  17 18,  19 20
```

would change the connectivities for element offsets 1, 3, 5, 7, and 9 in group 1. This must be a group of two noded elements (e.g., CROD), and these elements will now go between the node pairs 11-12, 13-14, 15-16, 17-18, and 19-20. Similarly the two lines:

```
CH   GR 5   EL   5001
1   2   3   4
```

would change the connectivity for the four-noded element numbered 5001 in group 5 to connect the nodes, 1-2-3-4.

The ADD command under the ELEMENT edit processor will add elements to a given group. The group is specified by the GROUP parameter. The ELEM and ID parameters are also valid. Sets of node numbers defining the connectivities of the element to be added are given on the line following the ADD command. For example if group 1 was composed of three noded CTRMEM elements, two new elements, at offsets 9 and 10 could be added to group 1 by using the following two lines:

```
ADD   GR   1   ID   9   10
5  6  7    15  16  17
```

The two new elements would be defined between nodes 5-6-7 and 15-16-17.

The HELP command will provide information on the valid material and property types for various element types or groups. The HELP ELEM command will list all of the elements supported by CADS by name. It also lists the property type name for the elements. The HELP command followed by either a specific material table name or property table name will define that table type. For example, HELP PR2 will state that the PR2 property type is for CONROD or CROD elements. HELP MATC would list the valid keywords for the composite material table.

The HELP PROP GROUP = number command will list the property type valid for the group given by the number after the GROUP parameter. If the HELP PROP TYPE = name command is used the property type for the given element type is listed.

The END command returns control to the EDIT module.

8.4 PROPERTY EDIT PROCESSOR

The PROPERTY processor under the EDIT module provides commands for modifying the current element size data. The PROPERTY prompt string is ? EDITPR while the syntax is:

```
LIST    GROUP
LIST    GROUP      number  ID     list
LIST    GROUP      number  EL     list
HELP    ELEM
HELP    PROPERTY   GROUP   list
HELP    PROPERTY   TYPE    name
CHANGE  GROUP      number  SIZE   NAME=value  EL   list
CHANGE  GROUP      number  SIZE   NAME=value  ID   list
END
```

The LIST command is the same as the LIST command described in 8.3 for the ELEMENT EDIT processor. The LIST command will list for the user: element, group, and type information. A detailed description is provided in 8.3.

The CHANGE command will change element sizes for the specified elements. The GROUP, ID, and ELEM keywords are valid for the CHANGE command. The number following the GROUP keywords defines the groups to be modified and the list following the ID parameter defines the element offsets in the groups which are to be changed. The ELEM keyword identifies a list of element numbers to be changed. The SIZE NAME parameter is replaced with an appropriate size parameter name for the group element type. This defines the actual data type to be changed for the elements.

For the SIZE NAME any of the valid size components for the type of elements in the change list may be used. These size component names are followed by the new size value for the elements with that component in the element list. For example,

```
CHANGE   GROUP  1  A = 1.25   ID  10 12 14
```

where group 1 is an axial rod (CROD) group with 50 elements, will make the cross-sectional area of elements 10, 12, and 14 equal to 1.25. The command

```
CH       GROUP  6  T = 1.5   EL  6001  6002
```

will change the thickness for the elements numbered 6001 and 6002 to 1.50.

The HELP command is the same as the HELP command described in 8.3 for the ELEMENT EDIT processor. The PROPERTY keyword requests the valid element size parameters for the given group or element type. The groups are specified using the GROUP parameter while the TYPE parameter indicates a specific element type for listing. For example the commands:

```
HELP   PROP   GR   1
HELP   PR     TY   CBAR
```

would first list the valid sizes available for group 1. The second command will list the valid size parameters for the CBAR element type.

The END command returns control to the EDIT module.

8.5 MATERIAL EDIT PROCESSOR

The MATERIAL processor under the EDIT module provides commands for modifying the current material property data. The MATERIAL prompt string is ? EDITMA while the syntax is:

```
LIST   GROUP
LIST   MAT. Type
ADD    MAT1=number            PARAMETERS=values
CHANGE GROUP=number           MAT1=number     ID    list
CHANGE GROUP=number           MAT1=number     EL    list
HELP   MAT Type
END
```

The LIST command is the same as the LIST command in 8.3, ELEMENT EDIT processor. It will list element or material information as requested. For example, the command

```
LIST   MAT1
```

will list the material values in the MAT1 tables. Similarly MAT2, MAT4 or MAT5 would be used to get the MAT2, MAT4, or MAT5 material tables. To list the layer composite material tables the keyword MATC is used.

The ADD command is used to add new material property values to tables for use in applying material values to the model. The syntax is ADD followed by the MAT1, MAT2, MAT4 or MAT5 keyword and then a list of parameters and values used to define the given table. These parameters are those valid for the particular table type and may be obtained using the HELP command. The material table defined by the ADD command should be completely described since it will go into the element data base as a complete table; that is, all of the added or new material parameters should be specified so that an invalid material is not stored. An example of the ADD command is:

```
ADD  MAT1  11  E = 10.6E6  U = 0.33
```

This command will add to the isotropic material numbered 11 a Young's Modulus of 10.6E6 and a Poisson's ratio of 0.33. The material tables defined by ADD are then used with the CHANGE command to change the material properties of specific lists of elements. The command:

ADD MAT2 2 A11 20.543E6 A33 6.7E6

will add the anisotropic material numbered 2 with the given A11 and A33 values.

The CHANGE command will change the material table callouts for a given list of groups and elements. The GROUP keyword defines the element group to be acted on. The MAT1 keyword is used to define the respective material table number to be applied to the given list of group and elements. The MAT1 keyword can be replaced by MAT2, MAT4, or MAT5 as needed. The EL keyword is used to define the list of element numbers whose materials are to be changed to the one given by the MAT1 parameter. The ID keyword defines a list of offsets within the GROUP whose materials are to be changed. For example to change group 1 element offsets 2 through 10 to the isotropic material table 5 use:

CHANGE GROUP 1 MAT1 5 ID 2 TO 10

The above command shows that the standard CADS TO/BY lists may be used for both the GROUP and ID lists, as well as the EL lists. Note that the CHANGE command cannot be used to change an element from one material type to another, i.e., from an isotropic MAT1 to a composite layer MATC type.

The HELP command will list out the valid keywords for the material property table type requested by the user, i.e., MAT1, MAT2, MAT4, MAT5, or MATC. Using MAT1 lists the valid isotropic material table keywords; using MAT2 lists the valid anisotropic material table keywords; using MAT4 or MAT5 the respective temperature Table 1 keywords are listed; and using MATC the composite layer table keywords are listed. It is the same as the HELP command in 8.3 ELEMENT EDIT processor.

The END command returns control to the EDIT module.

8.6 CASE EDIT PROCESSOR

The CASE processor under the EDIT module provides commands for modifying the current nongeometry data of the model; i.e., the analysis control or case control types of data. The CASE prompt string is ? EDITCA while the syntax is:

```
LIST     NASTRAN  n1  TO  n2  BY  n3
LIST     ANALYZE
LIST     OPTSTAT

INSERT   NASTRAN  n1
STOP

DELETE   NASTRAN  n1  TO  n2  BY  n3

REPLACE  NASTRAN  n1
REPLACE  ANALYZE
NR=n1    NSTR=n2
REPLACE  OPTSTAT
LMTDSP=n1  LSTCCL=n2  NR=n3  LPRINT=n4
END
```

The LIST command is used to list at the terminal the specified case control cards. The command format is: LIST followed by the analysis program name and an optional list of numbers. For example the command:

LIST ANALYZE

would list all of the ANALYZE non-geometric cards at the terminal. While the command:

LI NA 10 TO 20

would list the tenth through twentieth NASTRAN case control cards at the terminal. These numbers are used with the DELETE, INSERT, and REPLACE commands to specify which cards are to be modified.

The INSERT command is valid only with NASTRAN control decks. The format is: INSERT NASTRAN number. The number is the current case control line number after which new cards are to be inserted. This number is the current sequential position of the given card in the case control list and can be obtained by listing the cards at the terminal using the LIST command. New cards are entered until the STOP command is given to end the insertion of additional case control cards. Control returns to the CASE EDIT processor after STOP is entered.

The DELETE command is valid only with NASTRAN control cards. Its format is: DELETE NASTRAN number list. The number list is a standard TO/BY list of numbers which is used to specify the case control cards which are to be deleted from the list. The numbers are the sequential position numbers of the case control cards. For example the command:

DELETE NASTRAN 6

would delete card 6 from the case control card listing stored in the geometry data base. The command:

DELETE NASTRAN 10 TO 15

would delete cards 10 through 15 from the case control listing.

Note that ANALYZE and OPTSTAT do not have INSERT and DELETE command processing because these programs have few parameters which can be changed by the user. Changes to these values are performed by the REPLACE command. In addition, after a DELETE or INSERT command is processed the control deck is resequenced. Therefore, before performing another DELETE, INSERT, or REPLACE command the LIST NASTRAN command should be used to list the cards to be modified to insure that the correct card sequence numbers are specified.

Finally, the REPLACE command is used to replace ANALYZE and OPTSTAT values or NASTRAN control cards. For ANALYZE or OPTSTAT the format is: REPLACE followed by ANALYZE or OPTSTAT as appropriate. On the next line the parameters to be changed are specified followed by their new values. For instance the command:

REPLACE ANALYZE
NR=5 NSTR=3

would change the current ANALYZE values for NR and NSTR to 5 and 3, respectively. The commands:

```
REPLACE OPTSTAT
LMTDSP=3  LSTCCL=4  NR=5  LPRINT=6
```

will change the given OPTSTAT parameters to the specified values.

For NASTRAN the REPLACE format is: REPLACE NASTRAN number. The number is the line number which is to be replaced. After the REPLACE NASTRAN command the new case control line is entered. Thus, REPLACE replaces the entire case control line. For example:

```
REPLACE NASTRAN 10
$ THIS IS AN EXAMPLE
```

would replace the current control line 10 with the $ THIS IS AN EXAMPLE line.

The END command returns control to the EDIT module.

8.7 SAVE EDIT COMMAND

The SAVE command of the EDIT module is used to save the current geometric data base as a permanent file. It is essentially equivalent to SAVE in a typical system editor or the copy command at the system level. The command is executed by entering the word SAVE. The current internal headers will be updated and CADS will then copy each record from the current edited geometry data base on unit 11 to the current working geometry data base on unit 1.

When the EDIT module is entered the existing GEOMETRY data base is copied to a scratch data base on unit 11. The existing data base is the one attached to unit 1 and is specified during the CADS initialization procedure. It is specified by the answers to the initialization questions:

WILL YOU USE AN EXISTING DATA BASE (Y/N) ?

ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CASE OR END TO SKIP

ENTER THE TITLE FOR THE MODEL HEADER

ENTER THE NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP

During the editing process this data base is not modified until a SAVE command is executed in the EDIT module. At that time any changes made to the model are written to the GEOMETRY data base file specified in the initialization procedure and are available for display, output, or other operations.

3.8 COPY EDIT COMMAND

The COPY command of the EDIT module is used to copy a previously generated geometry data base to a new permanent file. The command is executed by entering the word COPY. CADS will then prompt for the new data set name to be used for copying the current geometry data base. This data set is then opened and attached to unit 12 and the current geometry data base is copied to it. The following prompt is used:

ENTER FILE NAME FOR EDITTED DATA TO BE COPIED TO

name (maximum of 40 characters)

The name is the filename to be used for the new copy of the geometry data base. The COPY command copies the GEOMETRY data base attached to unit 11 and so it is copying the data base file specified during the CADS initialization procedure with any changes made so far in the EDIT module.

9.0 POST OUTPUT TRANSLATOR

The CADS Post Output Translator (CADSPP) reads the output punch files from an analysis program and stores them to the POST data base. It currently stores element stress and force outputs as well as grid displacements and eigenvector values. The translator is a stand-alone program on the VAX 11/780 and thus must be executed separately before trying to perform deformed shape or stress value plots in CADS. The user commands for the translator are described in the following paragraphs.

The CADSPP commands are given interactively using the following prompting dialogue:

CADSPP asks for the file name of the analysis results. This is a card image file which is attached to unit 1. The prompt is:

ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP

the valid responses are:

end: to stop CADSPP
name: file name (max. 40 characters) containing card image data to be stored by CADSPP

Next the user is prompted for the type of analysis data by:

ENTER TYPE OF ANALYSIS DATA BEING STORED; VALID TYPES ARE:
NASTRAN, ANALYZE, OPTSTAT OR END TO STOP

the valid responses are:

end: to stop execution of CADSPP
nastran: NASTRAN analysis data is to be processed
analyze: ANALYZE analysis data is to be processed
optstat: OPTSTAT optimization data is to be processed

The user next specifies whether an existing post data base is to be updated or a new data base created and then the file name of the old or new post data base. The prompts from CADSPP are:

IS A NEW POST DATA BASE TO BE GENERATED (Y/N)?

where the responses are:

no: the post data base already exists and will be updated
yes: the post data base is being created with this run of CADSPP

The data set name is given by answering:

ENTER THE FILE NAME FOR THE POST DATA BASE

where the response is:

name: post data base name of up to 40 characters.

This name is checked against the answer to the existing post data base question. If the file name existence and existing data base answers do not match the user is prompted with these two questions again. This is to help prevent accidental damage to existing data sets.

If NASTRAN data is being processed a prompt to define the data format, either STATIC or DYNAMIC, of the input data is given:

ENTER INPUT TYPE STATIC OR DYNAMIC (STATIC/DYNAMIC)?

the valid responses are:

static: the standard static inputs are given
dynamic: the output results are dynamic, time history values

The STATIC/DYNAMIC question provides information on the format of the NASTRAN analysis program's card image output file. The command STATIC is used when the output is given without time step data. STATIC is the format of standard element stress and force data or grid-point displacements and eigenvector output by COSMIC NASTRAN under rigid formats 1 and 3. DYNAMIC is the standard output for rigid format 12. For DYNAMIC, output data is provided for each time step of a transient problem and can rapidly generate very large amounts of data for even a very simple model.

Finally the types of data blocks to be processed are requested by:

SPECIFY DATA BLOCK TYPES TO BE STORED; VALID NAMES ARE:
FORCE, STRESS, DISPLACE, EIGENVECTOR OR ALL

where the valid responses are:

force:	element forces are to be stored (NASTRAN only)
stress:	element stresses are to be stored (all programs)
displace:	node displacements are to be stored (all programs)
eigenvector:	mode shape values are to be stored (NASTRAN only)
all:	store all valid data blocks in the input data set

After these first three commands are given, a list of data types to be stored is specified by the user. The data types supported by the POST translator are STRESS, FORCE, DISPLACEMENT and EIGENVECTOR. These are the names of the element stresses or forces and grid displacements or eigenvectors, respectively. To specify which data blocks are to be stored, their respective names are entered as a command list on the line after the NASTRAN, OPTSTAT, or ANALYZE command.

The CADSPP translator program can store up to 60 load conditions per data file. If more than 60 cases are used for a model, additional post data bases would be required to store all of the cases.

A summary description of each command follows. In all cases the commands are entered in free format and the first two characters of the command may be used to abbreviate its name.

STATIC -	Typical output data format for NASTRAN information. Data is not transient.
DYNAMIC -	Transient analysis was performed and output results are given for multiple time steps. Program will store all data for each time step.
NASTRAN OPTSTAT ANALYZE	The analysis program formats accepted by CADSPP for storage on the POST data base. One name must be entered.
STRESS FORCE DISPLACE EIGENVECTOR ALL	Types of output data which may be stored by the POST translator. Keywords are entered on a single line using as many data types as desired by the user.

10.0 VAX EXECUTION PROCEDURES

10.1 CADS PROGRAM EXECUTION

The CADS software has been developed using the DEC VAX 11/780 and Tektronix 4014 hardware and the Precision Visuals, Inc., DI-3000 graphics package. The CADS software is coded in ANSI standard Fortran 77 without using the enhancements available in some Fortran 77 compilers. Preliminary compilations and limited testing indicate that the CADS code is very machine independent and will also operate on PRIME, CDC, and IBM processors, with few changes.

More specifically, for the DEC VAX 11/780 type of equipment the user must first log into the VAX VMS system using the logon and access procedures established for the particular installation. Once on the system, CADS is started using

RUN CADS

The RUN CADS command is used to begin execution of the actual CADS software.

This command assumes that the executable file is stored on the user's catalogue. More likely, the executable will be kept on a shared file for access by all users. In this case, the RUN command file name will be installation dependent.

The file defaults for the CADS program are Fortran units 1 through 20 for scratch files, standard terminal I/O units, and CADS message units. It is suggested that any user specified steering inputs for natural generation be assigned to file 21 or above. A short description of units 1-20 is presented in Table 9.

TABLE 9

UNIT DESCRIPTIONS FOR CADS RESERVED UNITS

UNIT	DESCRIPTION
1	Direct Access Geometry
2	Direct Access Geometry
3	Steering File Echo
4	Post File Direct Access
5	Card Image Terminal I/O
6	Terminal Messages from CADS
7	Terminal Error Messages from CADS
8	DI-3000 Debug Messages
9	DI-3000 Error Messages
10	Nastran Scratch File
11	Edit Save
12	Edit Copy file
13-19	Not Yet Used
20	Default Bulk Data Output Unit

10.2 CADSPP PROGRAM EXECUTION

The CADSPP program is a small, stand-alone program which translates an analysis program's output data into a format usable by the CADS program for further processing or display. CADSPP is written in standard Fortran 77 and, in initial testing, has been quickly transported between VAX 11/780 and IBM processors.

In order to run the CADSPP program the user must logon into the VAX system and have a card image output file from an analysis program available to CADSPP. CADSPP is executed using

RUN CADSPP

As with CADS, this assumes that the CADSPP executable file is stored on the individual user's catalogue and not on an installation dependent program library catalogue. CADSPP will prompt for the file names of the analysis program outputs and the direct access post data base.

11.0 ERROR MESSAGES

The CADS software will attempt to recover from input or processing errors in one of several different ways. The type and severity of the error will define the error handling procedure to be used by CADS.

The most common errors are generally mistypings of command words, options, or parameters. In these cases, CADS will say that particular option or command was not found or is not valid and ask that the entire command line be re-entered. The user should then enter the entire line with the correct spellings and options and the software will continue processing from that point.

CADS will check parameter numeric values for real or integer numbers as required. If an incorrect or mistyped numeric value is entered, CADS will echo the character string and request that a real or integer number be entered. In this case, the user should enter the required numeric value only and not the entire command line. CADS will then use that value in the command and continue processing.

Finally, the DI-3000 graphics package may issue a warning or error message based upon some series of actions it is taking. The level at which errors will be printed out and the unit to which they will be printed can be changed by the CADS software maintenance personnel. The JSETER and JFILES routines control the DI-3000 error messages. Typically, DI-3000 will continue processing after an error message through its own internal routines. The CADS command may have to be re-entered and/or modified to obtain a correct display after a DI-3000 error message, since DI-3000 may not have taken the appropriate action in processing the given error.

12.0 SAMPLE SESSIONS

12.1 BACKGROUND

The objective of this section is to provide a series of user inputs, short descriptions, and program outputs to illustrate the functions of the CADS software. The data used to make up these samples is provided as files on the magnetic tape containing the CADS software. The tape is described in Section 2.0 Installation, of the Program Maintenance Manual, Volume III of the CADS final report.

It should be remembered that these are just examples of various CADS command uses, operations and sequences. A wide variety of other examples or command sequences could be used in other applications. In general the full name, instead of the abbreviation, for a command or keyword was used in these samples in order to simplify the reading of the material. The format for these examples is a series of commands on the left half of the page with descriptive comments on the right side. This page is followed by the terminal display of the output.

These sessions were run on a DEC VAX 11/785 processor using a Tektronix 4014 terminal and a Tektronix hardcopy device. All inputs and outputs are in uppercase characters since these sessions are copies of the actual screen displays.

12.2 ANALYZE TEST CASE

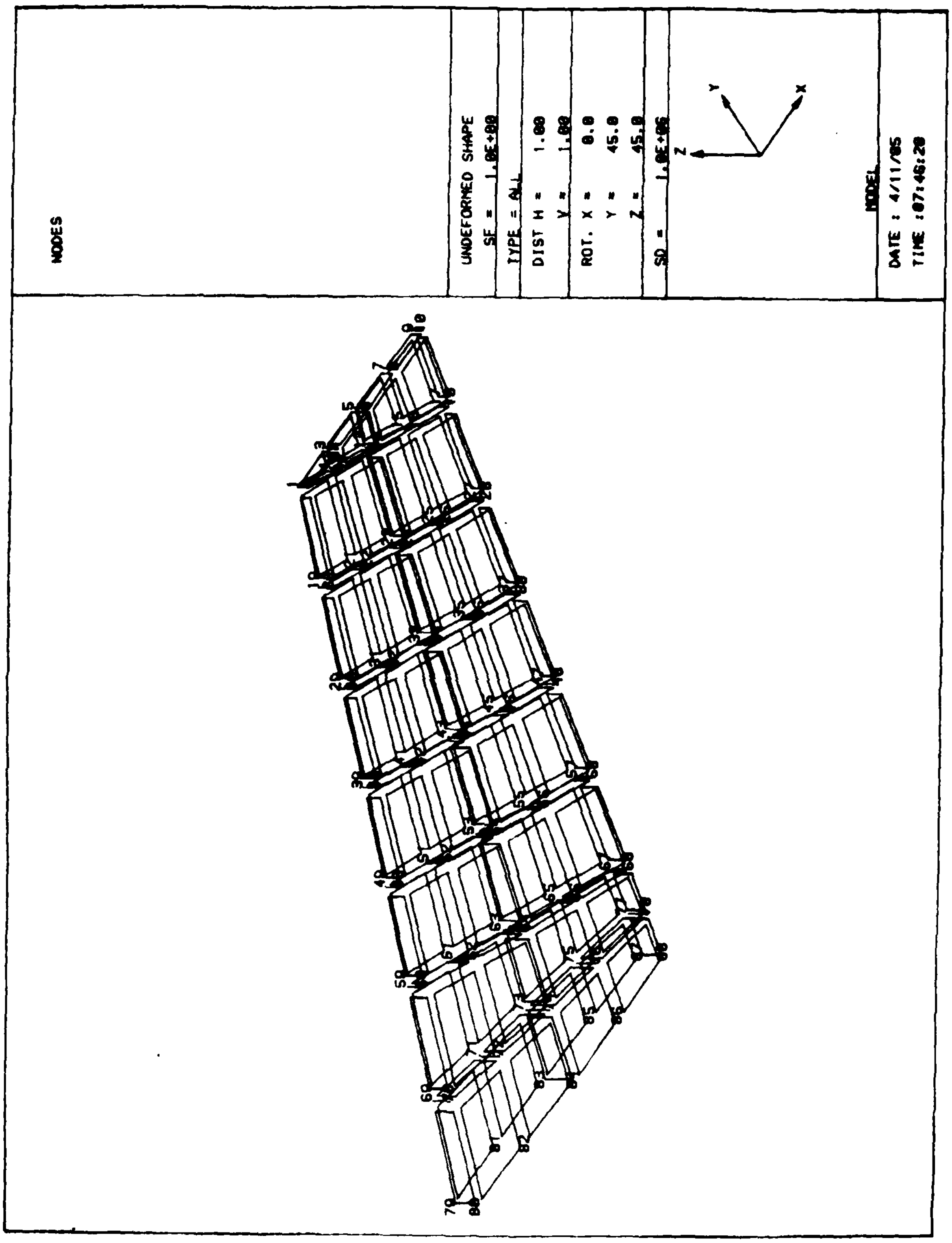
This case is an intermediate complexity wing of isotropic materials. Its purpose is to demonstrate commands or keywords specific to ANALYZE. These are primarily the BEGIN ANALYZE command in the READ module and the element stress component names under the ATTRIBUTE submodule of DISPLAY.

```
RUN CADS
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

ANALYZE
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
Y
 ENTER POST DATA BASE FILE NAME FOR CADS PLOTTING OR END TO SKIP
POSTANAL.DAT
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
N
 ENTER THE TITLE TO MODEL HEADER
GEOMETRY DATA FOR ANALYZE
 ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP
GEOMANAL.DAT
 ? CADS

READ
 ? READ

BEGIN ANALYZE
 ENTER ANALYZE INPUT FILE NAME NOW OR END TO RETURN
ANALIN.DAT
      4  ACCEPTABLE GROUPS PROCESSED

  GROUP    1    CROD    = 2          39  ELEMENTS
  GROUP    2    CTRMEM  = 3           2  ELEMENTS
  GROUP    3    CQDMEM2 = 4          62  ELEMENTS
  GROUP    4    CSHEAR  = 5          55  ELEMENTS
 ? READ

END
 MODULE READ     ENDED. TIME =07:44:36  DELTA =   64.96
 ? CADS

SET
 ? SET

E1 ALL
 ? SET

PLOT E1
```

This is the start of the ANALYZE input data deck example. The RUN CADS command was given on a Tektronix 4014 type terminal under the DEC VAX VMS 3.7 operating system to begin CADS. The terminal type, baud rate, communication type (ANALYZE), and data base questions were then answered. Note a POST data base was in existence but the GEOMETRY data base was not; normally a GEOMETRY data base would be in existence before the POST data base. The ANALYZE card data was read in under the READ module and an element set of all the elements, called E1, was made. This set was sent to the DISPLAY module using the command PLOT E1.

```
? DISPLAY
ROTATE Y 45.0  Z 45.0
 ? DISPLAY
PLOT NODE BREAK
```

The model was rotated 45.0 degrees about the Y and Z axes using the ROTATE command. It was then plotted using the PLOT command with the NODE and BREAK keywords to plot the node numbers and the elements individually. The result is shown on the next page.



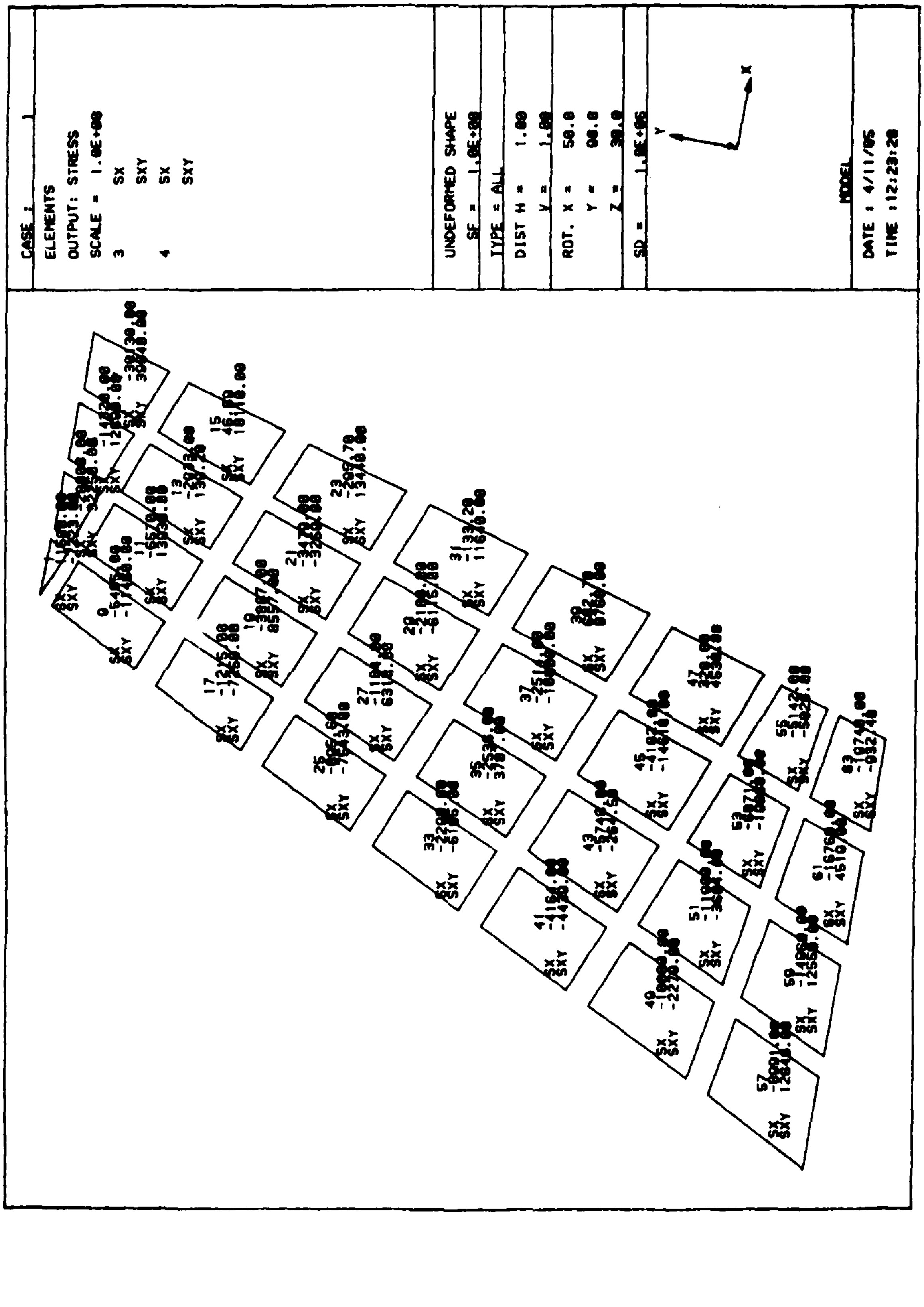
NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :07:46:20

```
 ? DISPLAY
ROTATE  X 50.0  Y 90.0  Z 30.0
 ? DISPLAY
DEFINE ID 1 TO 63 BY 2
```

A new rotation is specified which changes the model display angles to 50.0, 90.0, and 30.0 degrees about the X, Y, and Z axes, respectively. Next the DEFINE command is used to define a new element set for plotting. This set is composed of the odd numbered ANALYZE elements from 1 to 63. The screen is then cleared in preparation for additional plotting.

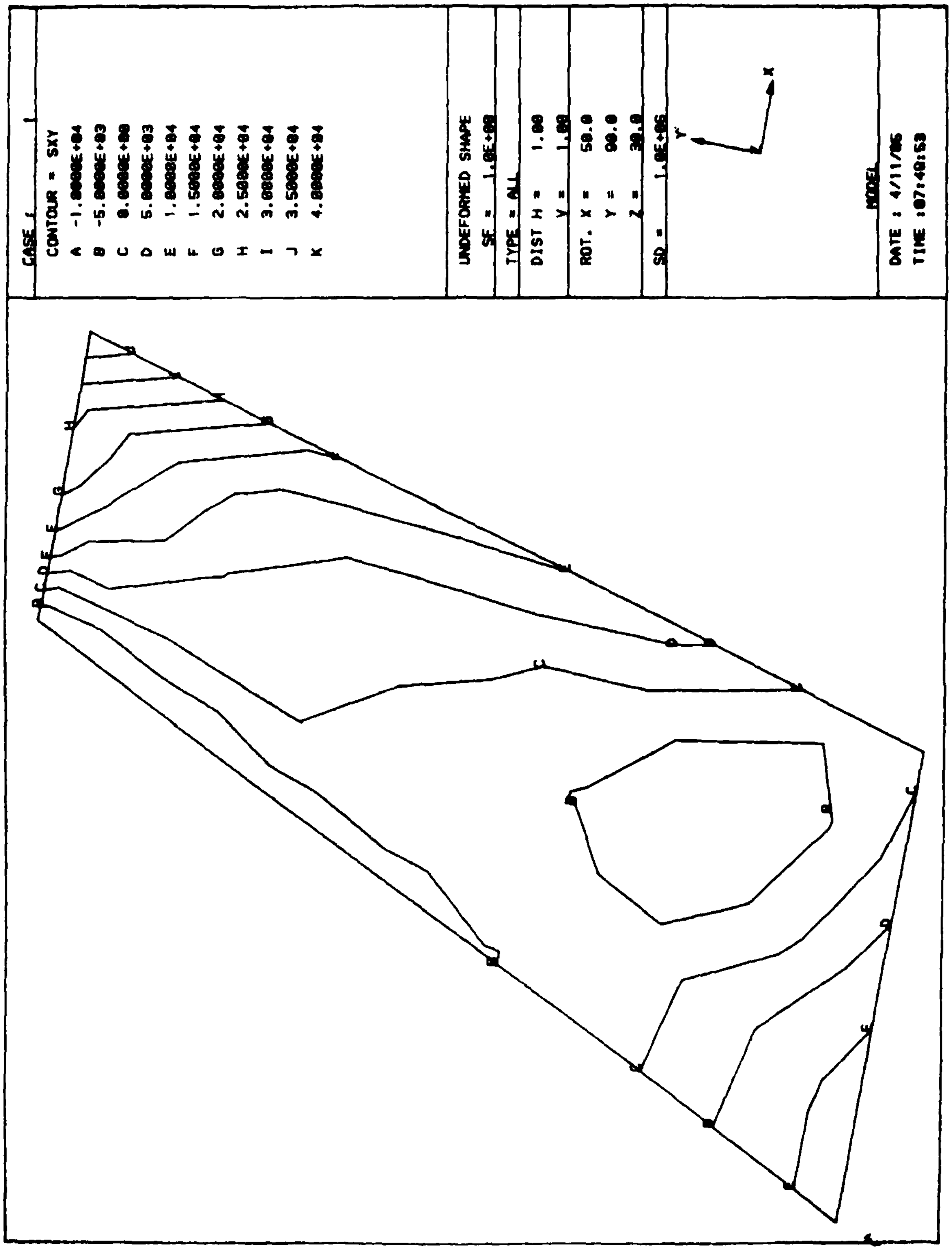
```
 ? DISPLAY

CASE 1
 ? DISPLAY

BEGIN ATTRIB
 ? ATTRIB

PROGRAM ANALYZE
 ? ATTRIB

MODE STRESS
 ? ATTRIB

HELP 3
 STRESS   COMPONENTS OF ELEMENT TYPE 3          ARE
 SX     SY     SXY   EFS1   ENER   MS
 ? ATTRIB

HELP 4
 STRESS   COMPONENTS OF ELEMENT TYPE 4          ARE
 SX     SY     SXY   EFS1   EFS2  EFS3  EFS4  ENER  MS
 ? ATTRIB

3 SX SXY
 ? ATTRIB

4 SX SXY
 ? ATTRIB

END
 MODULE ATTRIB   ENDED, TIME =07:47:50  DELTA =  203.10
 ? DISPLAY

PLOT BREAK ELEMENT STRESS
```

This command sequence illustrates the definition of an analysis result case as well as the specification of element stress components for display. The CASE command specifies the output case 1; while the BEGIN ATTRIB command starts the stress component definition. Since the ANALYZE communication type is being used, element names are given using 3 and 4 which are the ANALYZE names for the triangular and quadrilateral membrane elements. The element numbers, SX, and SXY stresses are shown on the next plot.


CASE : 1
ELEMENTS
OUTPUT: STRESS
SCALE = 1.0E+00
3 SX
SXY
4 SX
SXY
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 50.0
Y = 00.0
Z = 30.0
SD = 1.0E+06
Y
X
MODEL
DATE : 4/11/85
TIME :12:23:20

```
 ? DISPLAY

BEGIN ATTRIB
 ? ATTRIB

PROGRAM ANALYZE
 ? ATTRIB

MODE STRESS
 ? ATTRIB

3 SXY
 ? ATTRIB

4 SXY
 ? ATTRIB

END
 MODULE ATTRIB   ENDED, TIME =07:40:32  DELTA =   03.47
 ? DISPLAY

PLOT CONTOUR STRESS
```

The ATTRIBUTE module is re-entered and the SXY stress component is specified for the type 3 and 4 elements. These stresses are then displayed as contour lines using the PLOT command with the CONTOUR and STRESS keywords. Note that the PROGRAM name and MODE type are required before specifying the element components.



CASE 1
CONTOUR = SXY
A -1.0000E+04
B -5.0000E+03
C 0.0000E+00
D 5.0000E+03
E 1.0000E+04
F 1.5000E+04
G 2.0000E+04
H 2.5000E+04
I 3.0000E+04
J 3.5000E+04
K 4.0000E+04
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 50.0
Y = 90.0
Z = 30.0
SD = 1.0E+06
Y
X
MODEL
DATE : 4/11/86
TIME :07:40:53

12.3 OPTSTAT TEST CASE

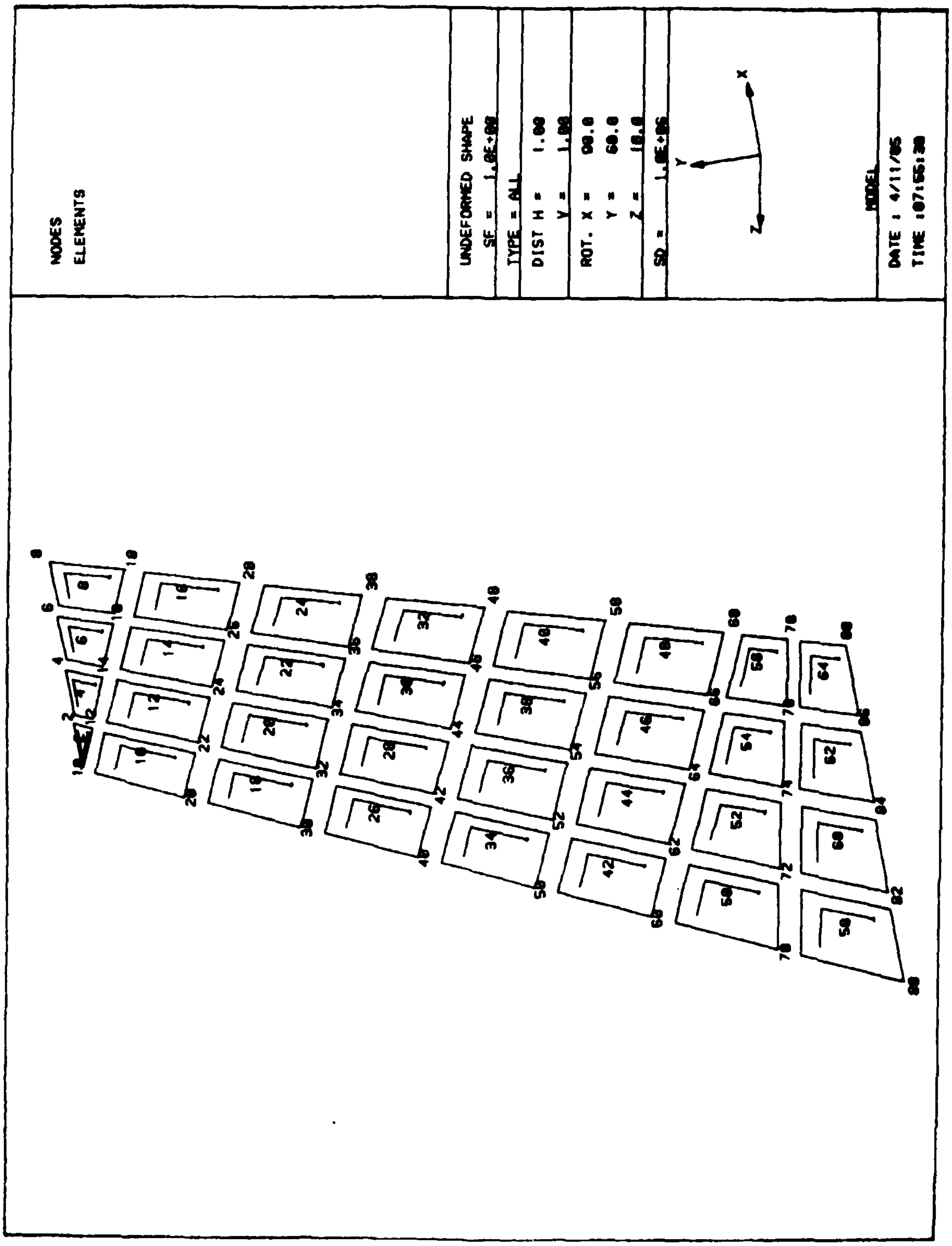
This case is an intermediate complexity wing of composite materials. Its purpose is to demonstrate commands or keywords specific to OPTSTAT. These are primarily the BEGIN OPTSTAT command in the READ module and the element stress component names under the ATTRIBUTE submodule of DISPLAY.

```
 ? CADS

END
 MODULE CADS      ENDED. TIME =07:50:14  DELTA =    3.90
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
YES
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

OPTSTAT
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
Y
 ENTER POST DATA BASE FILE NAME FOR CADS PLOTTING OR END TO SKIP
POSTOPT.DAT
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
N
 ENTER THE TITLE TO MODEL HEADER
GEOMETRY FOR OPTSTAT
 ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP
GEOMOPT.DAT
 ? CADS

READ
 ? READ

BEGIN OPTSTAT
 ENTER OPTSTAT INPUT FILE NAME NOW OR END TO RETURN
OPTIN.DAT
      4  ACCEPTABLE GROUPS PROCESSED

  GROUP   1    CROD    = 2          30  ELEMENTS
  GROUP   2    CTRMEM  = 3           2  ELEMENTS
  GROUP   3    CODMEM2 = 4          62  ELEMENTS
  GROUP   4    CSHEAR  = 5          55  ELEMENTS
 ? READ

END
 MODULE READ      ENDED. TIME =07:51:23  DELTA =   61.50
 ? CADS

SET
 ? SET

E1 ID  2  TO  64  BY  2
 ? SET

PLOT E1
```

The ANALYZE example is then ended and CADS asks to start a new model. A yes (Y) was given so that the OPTSTAT test case could be processed. It is still a 4014 terminal at 9600 baud, but now CADS is using the OPTSTAT communications mode and POSTOPT.DAT analysis results. A new geometry data base is specified as GEOMOPT.DAT, and the OPTSTAT data was read in using the BEGIN OPTSTAT command in the READ module. Finally, an element set called E1 containing elements numbered from 2 to 64 by 2 was specified and sent to the DISPLAY module.

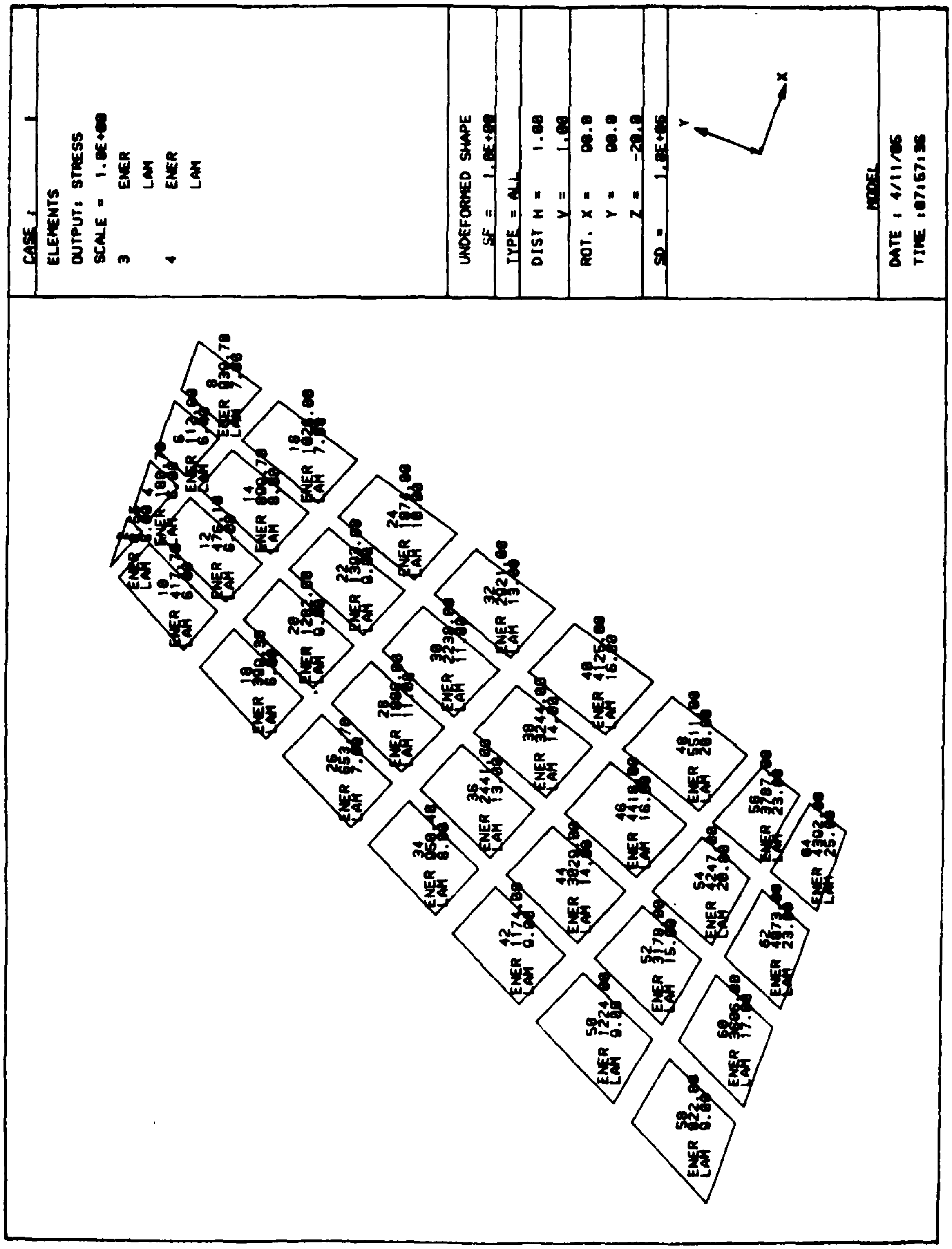
```
 ? DISPLAY
ROTATE  X 90.0  Y 60.0  Z 10.0
 ? DISPLAY
PLOT BREAK NODE ELEMENT AXIS
```

The element set was rotated about the X, Y, and Z axes using the ROTATE command and plotted using the PLOT command. The plot keywords BREAK, NODE, ELEMENT, and AXIS are used to show the elements individually with the node and element numbers and the element local axis information. The plot is on the next page.


NODES
ELEMENTS
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 90.0
Y = 60.0
Z = 10.0
SD = 1.0E+06
MODEL
DATE : 4/11/85
TIME :07:55:30

```
? DISPLAY

ROTATE  Z -20  Y 90
 ? DISPLAY

BEGIN ATTRIB
 ? ATTRIB

PROGRAM OPTSTAT
 ? ATTRIB

MODE STRESS
 ? ATTRIB

HELP 2
 STRESS   COMPONENTS OF ELEMENT TYPE 2        ARE
 SX    EFS   ALS1  ALS2  ENER  OPTT
 ? ATTRIB

HELP 3
 STRESS   COMPONENTS OF ELEMENT TYPE 3        ARE
 SX    SY    SXY   EFS   ALS1  ALS2  ALS3  ALS4  ALS5  ENER
 OPTT  LAM   THK8  AEX   THK9  AEY
 ? ATTRIB

HELP 4
 STRESS   COMPONENTS OF ELEMENT TYPE 4        ARE
 SX    SY    SXY   EFS   ALS1  ALS2  ALS3  ALS4  ALS5  ENER
 OPTT  LAM   THK8  AEX   THK9  AEY
 ? ATTRIB

HELP 5
 STRESS   COMPONENTS OF ELEMENT TYPE 5        ARE
 SXY   EFS   ALS1  ALS2  ALS3  ALS4  ALS5  ENER  OPTT
 ? ATTRIB

3 ENER LAM
 ? ATTRIB

4 ENER LAM
 ? ATTRIB

END
 MODULE ATTRIB   ENDED, TIME =13:46:16  DELTA =   83.63
 ? DISPLAY

CASE 1
 ? DISPLAY

PLOT ELEMENT BREAK STRESS
```

The plot was rotated and the ATTRIBUTE module was begun to specify the OPTSTAT output data from the POST database. The element STRESS mode was set by the MODE command and the HELP command was used to get the valid component names for the OPTSTAT element types. Next the ENERGY and LAM stress components were specified for element types 3 and 4 and ATTRIBUTE was ended. LAM is the optimized number of layers for the element as determined by OPTSTAT. For non-composite elements the optimized size is obtained by the OPTT component name. The CASE 1 command defines the load condition for display and the PLOT command was issued.


CASE : 1
ELEMENTS
OUTPUT: STRESS
SCALE = 1.0E+00
3 ENER
LAM
4 ENER
LAM
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 90.0
Y = 90.0
Z = -20.0
SD = 1.0E+06
Y
X
MODEL
DATE : 4/11/86
TIME : 07:57:36

12.4 NASTRAN SOLID TEST CASE

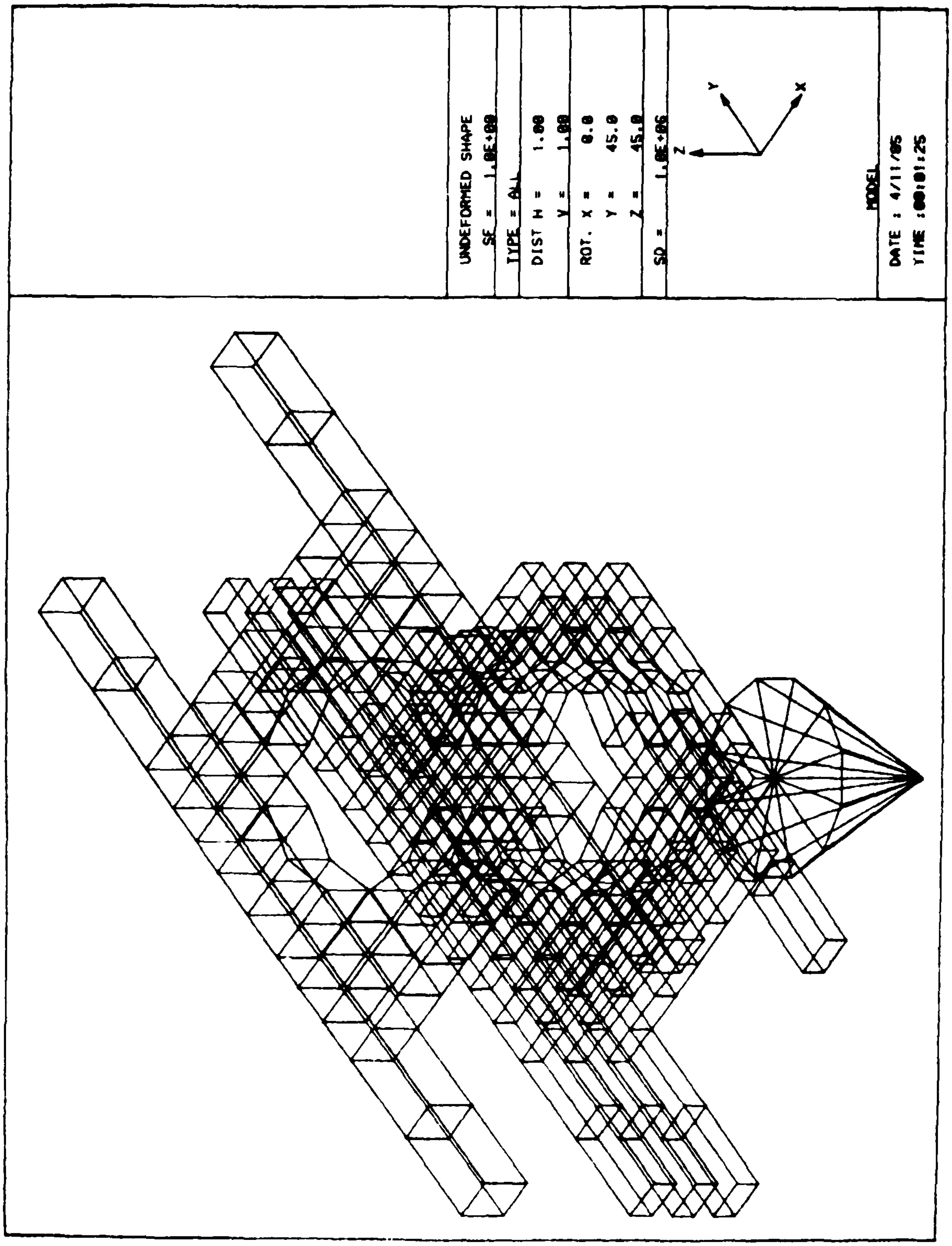
This case is a solid element model of a metal fitting. It is fairly complicated and demonstrates the use of the CADS display commands for a solid model. It is a NASTRAN model with the element connectivities, grid point locations, and material properties defined.

```
? CADS

END
 MODULE CADS     ENDED, TIME =07:50:20  DELTA =    3.57
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
Y
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NASTRAN
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
Y
 ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP
GEOMNAST1.DAT
 ? CADS

SET
 ? SET

LIST GROUP
  GROUP    1    CONROD  = CONROD     124  ELEMENTS
  GROUP    2    CELAS2  = CELAS1     144  ELEMENTS
  GROUP    3    CTETRA  = CTETRA      12  ELEMENTS
  GROUP    4    CWEDGE  = CWEDGE     224  ELEMENTS
  GROUP    5    CIHEX1  = CIHEX1     180  ELEMENTS
 ? SET

E1 GROUP 5
 ? SET

E2 CTETRA
 ? SET

E1 E1 U E2
 ? SET

PLOT E1
```

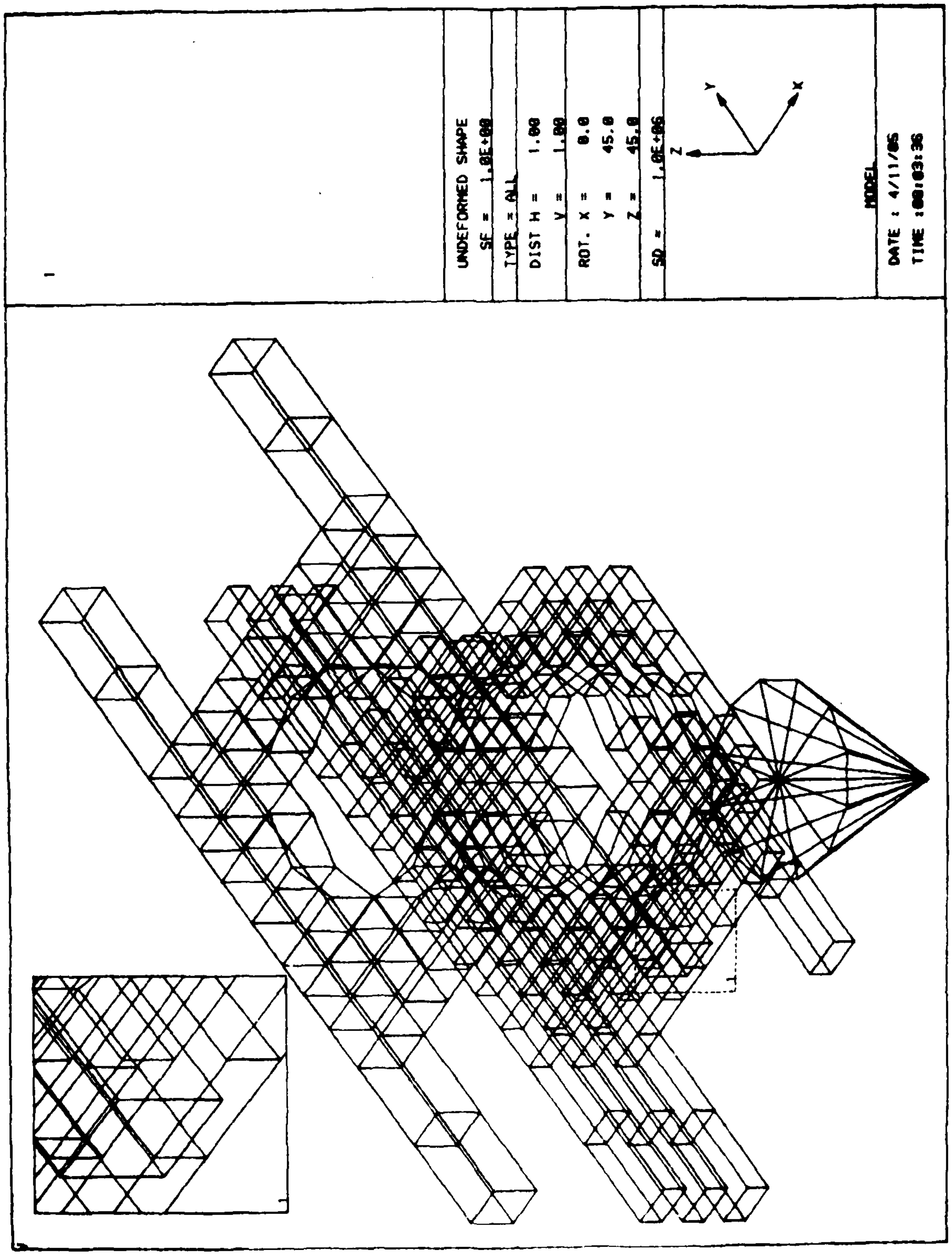
The program was ended and a new model was specified. This was a NASTRAN model without a POST data base but with an exisiting GEOMETRY data base. Therefore, GEOMNAST1.DAT was attached and the SET module was entered. The LIST GROUP command listed the model groups and the element sets were defined. Set E1 was group 5 and set E2 was the CTETRA elements. These two sets were combined as a union (U) using the E1 E1 U E2 command. This new E1 set was sent to the display module using PLOT E1.

```
? DISPLAY
ROTATE Y 45.  Z 45
 ? DISPLAY
PLOT
```

The element data was rotated 45.0 degrees about Y and Z. Note the 45 without the decimal point for the Z parameter. The ROTATE command knows that the rotations are to be real numbers and converts the integer 45 to a real 45.0 for the user. The PLOT command was issued to build the display shown on the next page.


UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :00:01:25

Once this display was drawn a V character was entered to illustrate the end of plot processing using the VIEW command. The V started the cursors so that the dashed line box in the lower left quadrant could be specified. Another box was defined in the upper left quadrant and an R character was entered. This caused CADS to redraw the elements in the dashed box into the solid line box in the upper corner.



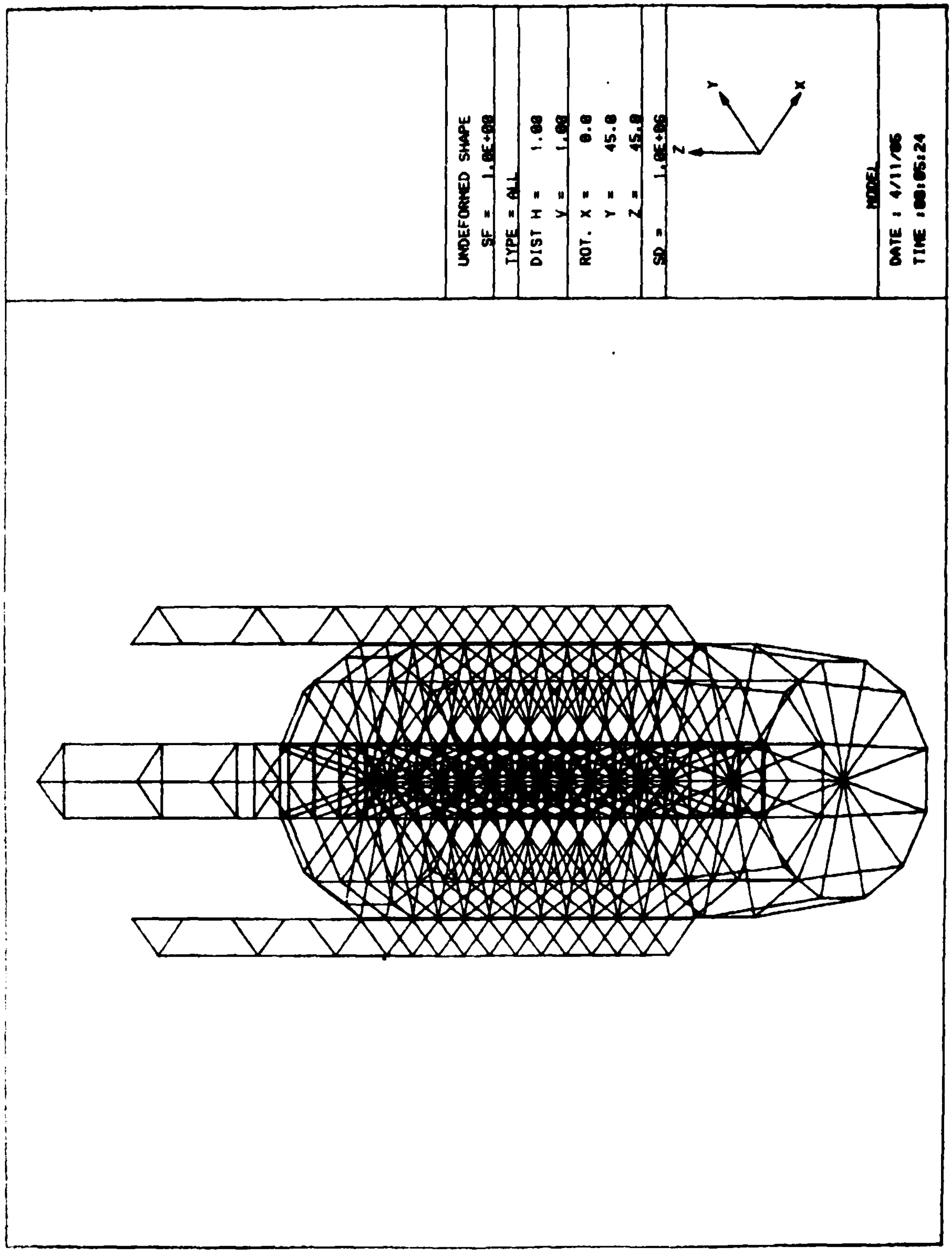
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :00:03:36

```
? DISPLAY
DEFINE CWEDGE
```

The display was completed and a blank character was entered to return to the DISPLAY module where a new element set was defined. This set is composed of all of the CWEDGE elements in the model. Note that since the NASTRAN communications mode is being used the element type names are specified using the appropriate NASTRAN element name. Again the screen is cleared.

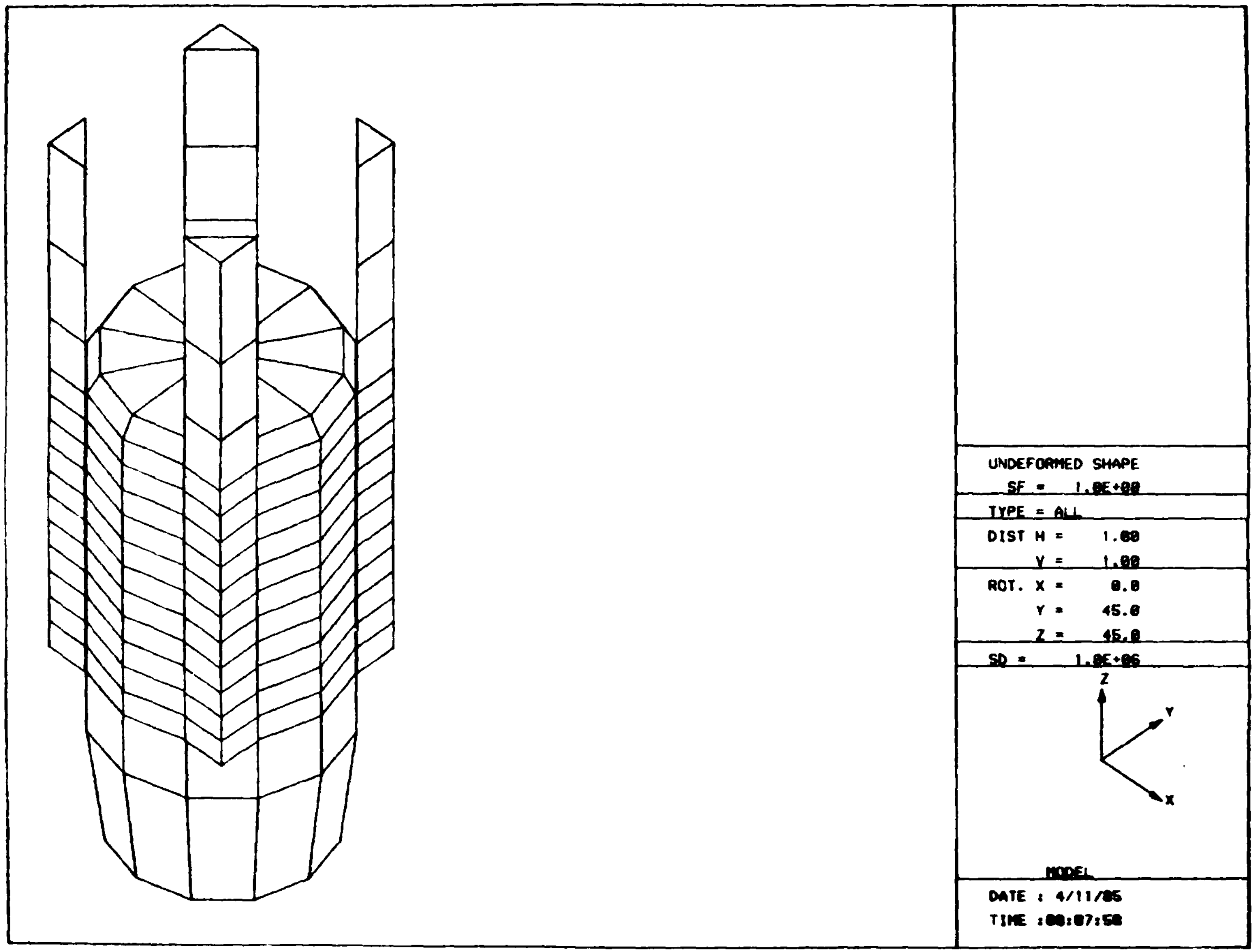
```
? DISPLAY
PLOT
```

The PLOT command is issued and the elements are plotted. Note that the BREAK keyword is not being used, so the default element display is used. This display does not show each element separately; instead the elements are drawn with common boundaries.


UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME :08:05:24

```
? DISPLAY
PLOT HIDE
```

The HIDE keyword is now specified on the PLOT command. This starts the hidden line processing for this particular display. The hidden line plot is shown on the next page.

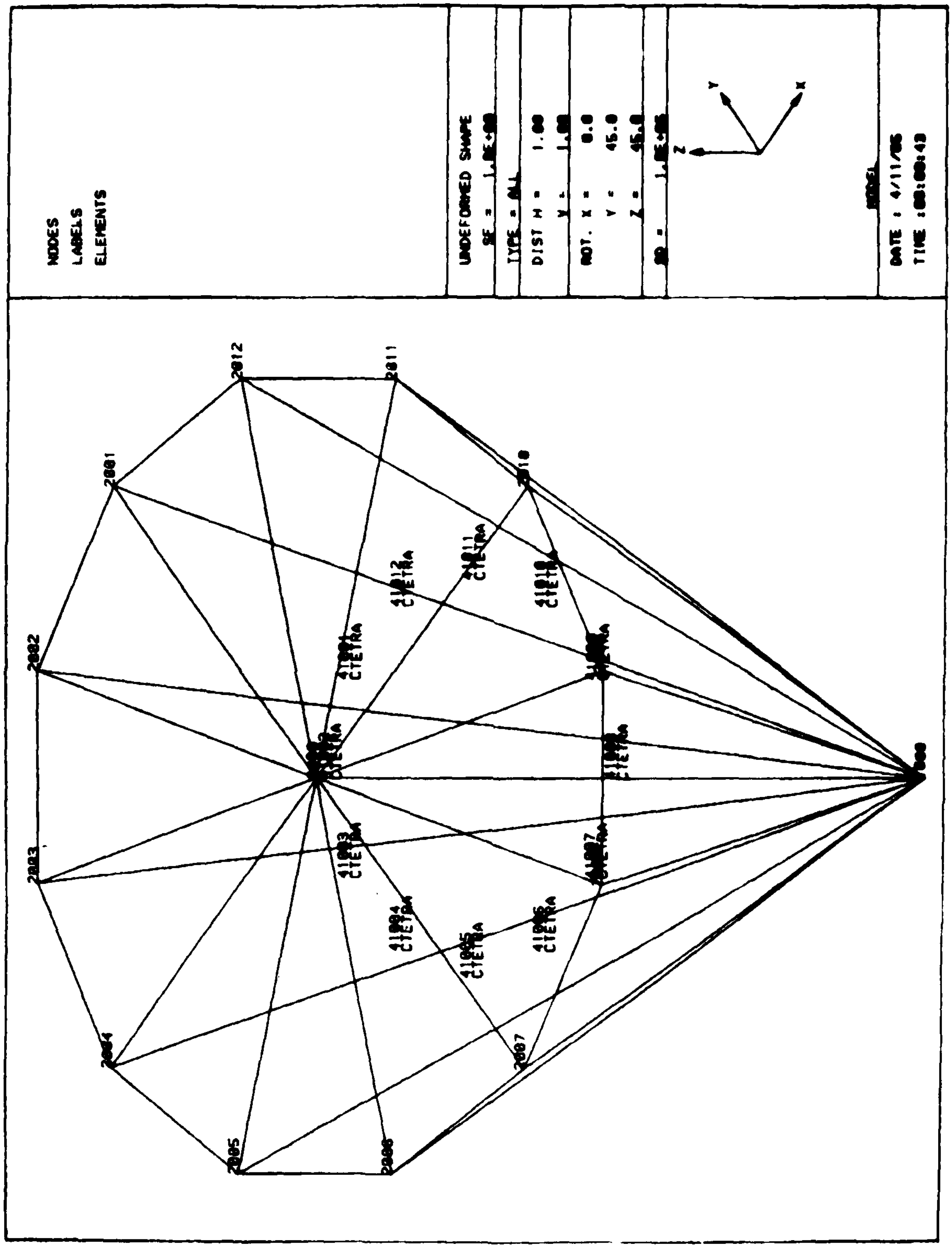

UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:07:58

```
? DISPLAY
DEFINE CTETRA
```

Another element set is defined for plotting. It is all of the CTETRA elements as shown by the DEFINE command.

```
? DISPLAY
PLOT NODE ELEMENT LABEL
```

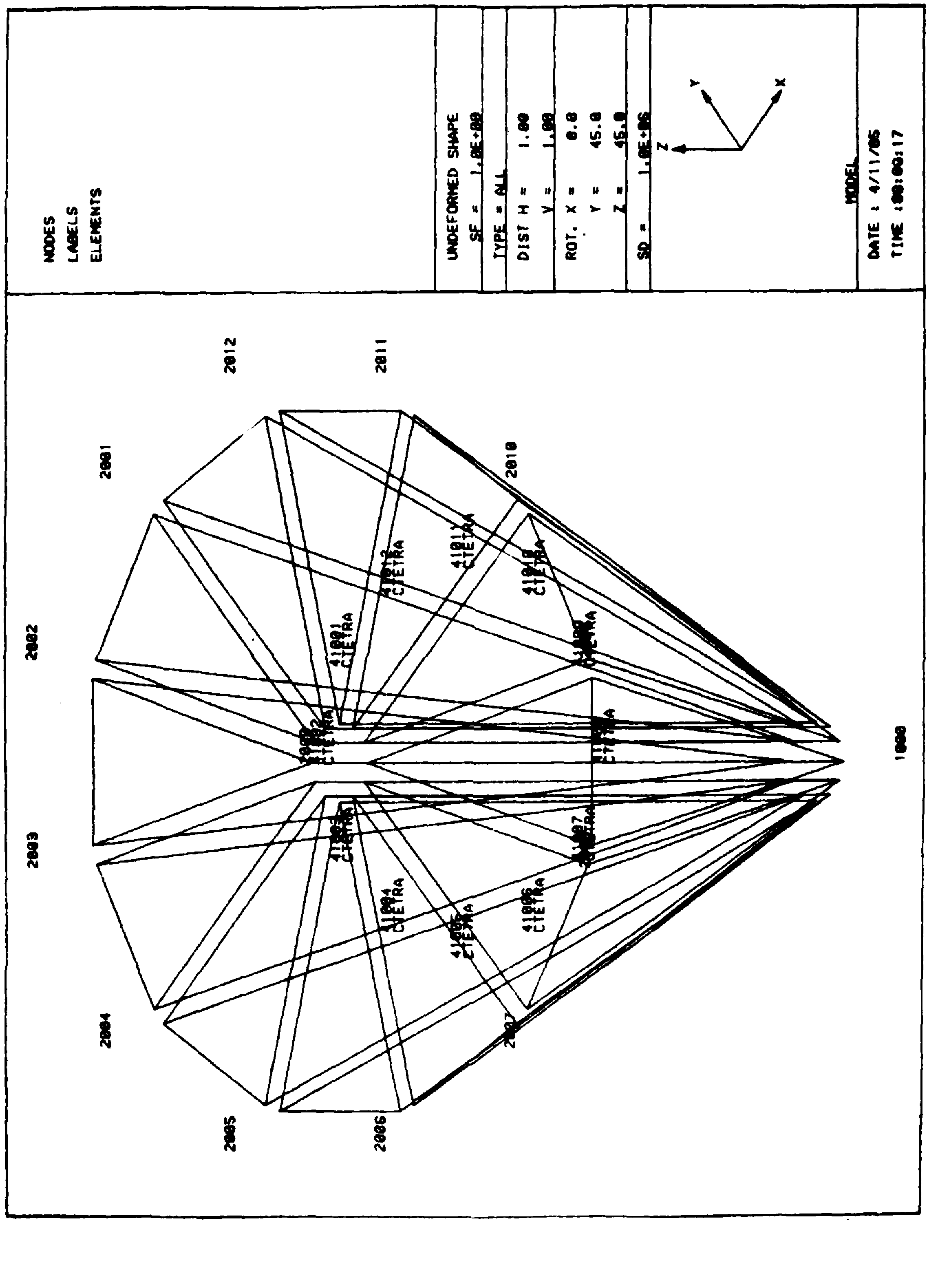
This new set is now plotted using the PLOT command with the NODE, ELEMENT, and LABEL keywords. This results in the display on the next page with the node numbers, element numbers and element labels (type name) plotted.



NODES
LABELS
ELEMENTS
UNDEFORMED SHAPE
TYPE = ALL
ROT. X = 0.0
Y = 45.0
2001
2002
2003
2004
2005
2006
2007
2011
2012
41004
CTETRA
41005
CTETRA
41006
CTETRA
41012
CTETRA
41011
CTETRA
41010
CTETRA

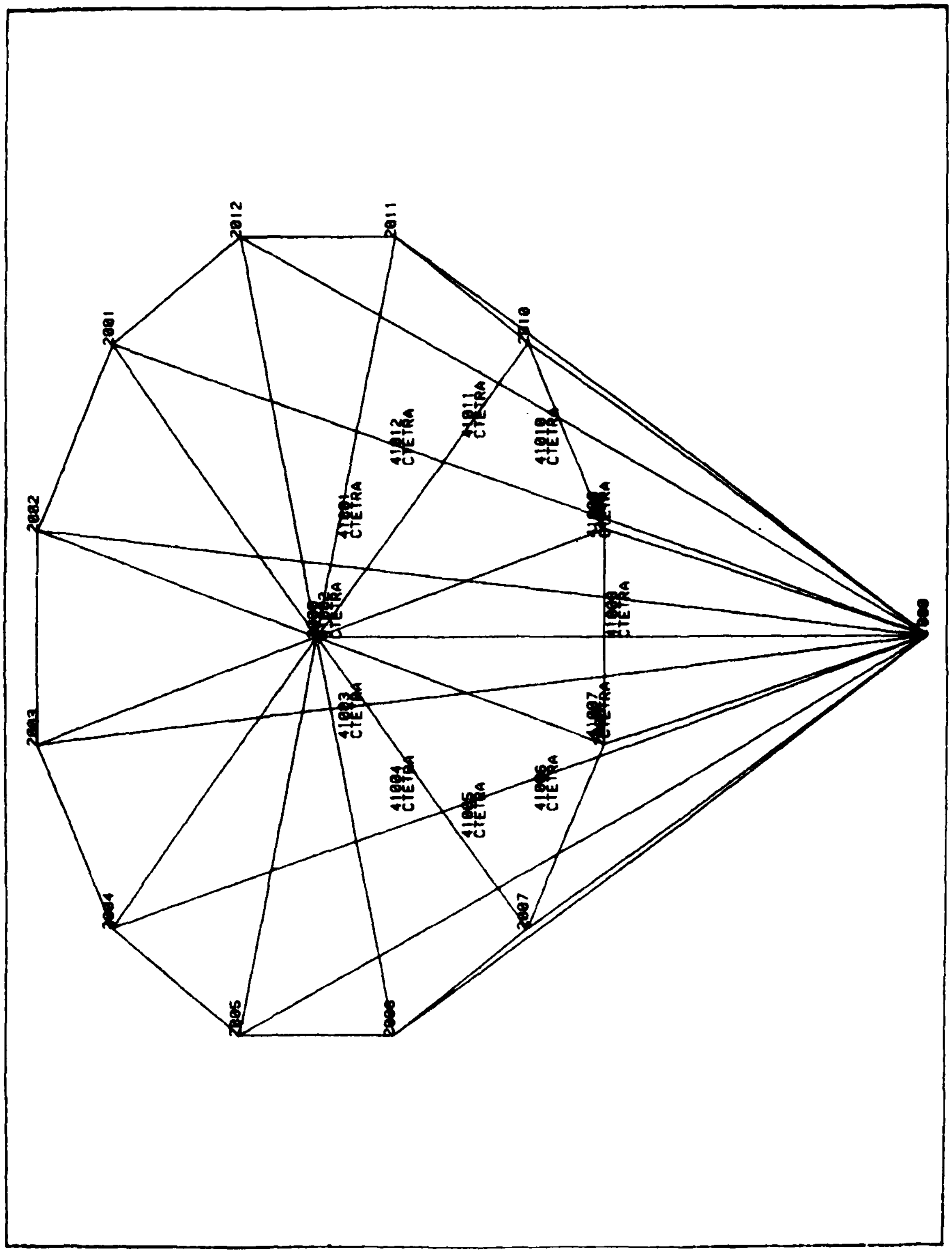
```
? DISPLAY
PLOT NODE ELEMENT LABEL BREAK
```

This PLOT command is the same as the previous one except it has added the BREAK keyword. This shrinks the elements about their centroids and results in the display on the next page. Note the margin information with the nodes, labels, and elements legend as well as the standard scale factor, type, distortion factor, rotations, axes, and date/time data.


NODES
LABELS
ELEMENTS
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :00:00:17
2001
2002
2003
2004
2005
2006
2010
2011
2012
1000
41001 CTETRA
41012 CTETRA
41011 CTETRA
41010 CTETRA
41004 CTETRA
41006 CTETRA

```
? DISPLAY
NOMARGIN
? DISPLAY
PLOT NODE ELEMENT LABEL
```

The NOMARGIN command turns off the default margin processing so that the display can be mapped to the entire terminal screen. The margin information will remain off now until a MARGIN command is given at a later time. The same plot with the element, node, and label information is shown.


2012
2011
2010
2001
2002
2003
2004
2005
2006
2007
41012 CTETRA
41011 CTETRA
41010 CTETRA
41001 CTETRA
41008 CTETRA
41003 CTETRA
41004 CTETRA
41005 CTETRA
41006 CTETRA
41007 CTETRA

```
? CADS
END
 MODULE CADS     ENDED, TIME =08:18:23  DELTA =     4.49
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
N
FORTRAN STOP
$
```

The DISPLAY module is ended and control is returned to the executive level. CADS is then ended using END, and since another model is not desired the no (N) response is made to the question:

DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?

12.5 NASTRAN WING TEST CASE

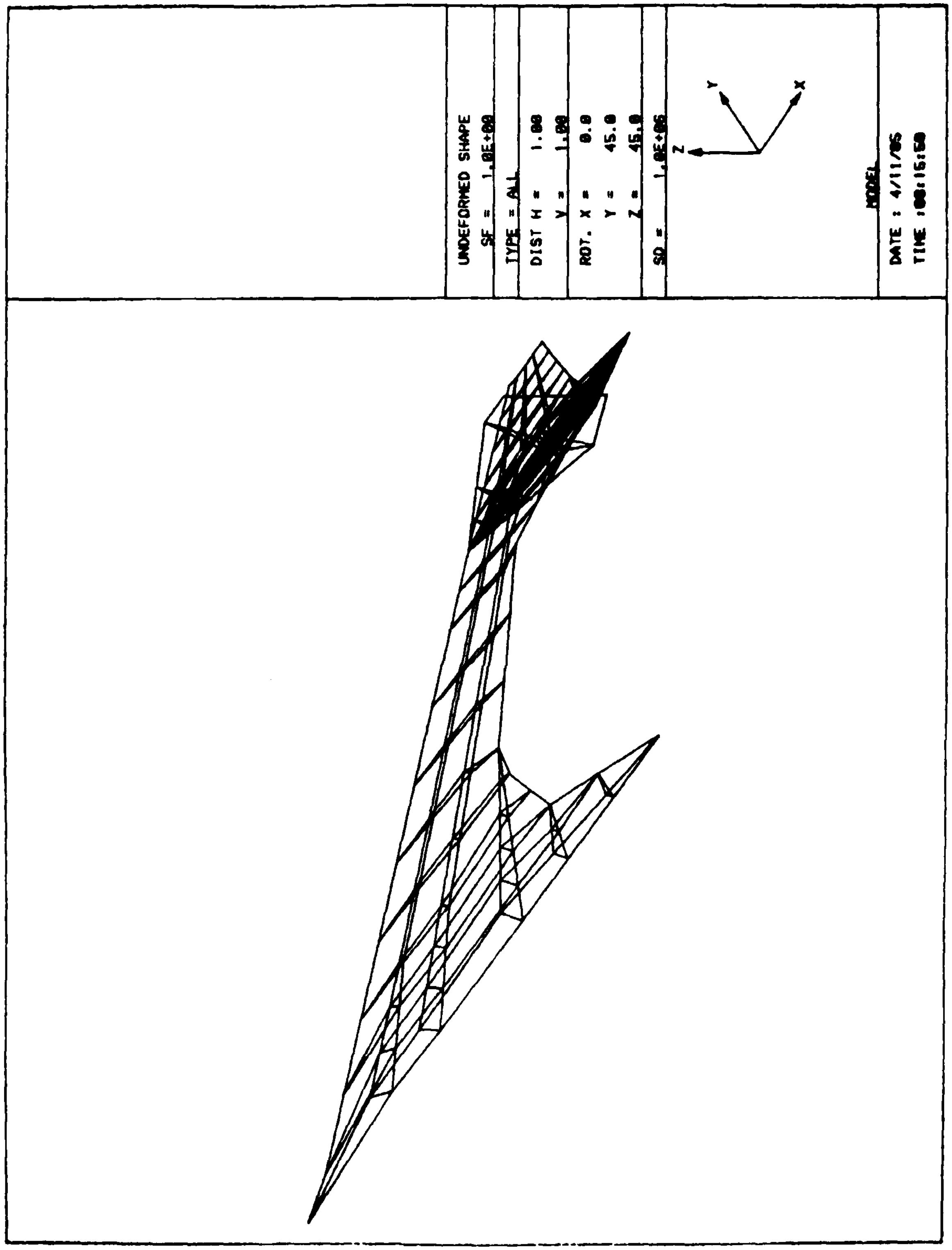
This case is a wing model developed by the Air Force Flight Dynamics Laboratory. It is used to demonstrate various DISPLAY module commands for investigating an existing finite element model. This model contains all of the NASTRAN input data required for execution from the Executive and Case control decks through the Bulk data input cards.

```
RUN CADS
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NASTRAN
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
N
 ENTER THE TITLE TO MODEL HEADER
TEST WING GEOMETRY
 ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP
GEOMNAST2.DAT
 ? CADS

READ
 ? READ

BEGIN NASTRAN
 ENTER NASTRAN INPUT FILE NAME NOW OR END TO RETURN
NASTIN2.DAT
 NUMBER OF ACCEPTABLE NASTRAN CARDS PROCESSED

    CQDMEM2      165
    PQDMEM2        2
    CROD         451
    PROD           2
    CSHEAR        96
    PSHEAR         2
    CTRMEM        96
    PTRMEM         2
    MAT1           1
    GRID         183
    GRDSET         1
    SPC1           1
 GROUP     1  CQDMEM2
 GROUP     2  CROD
 GROUP     3  CSHEAR
 GROUP     4  CTRMEM
      4  ACCEPTABLE GROUPS PROCESSED

  GROUP     1    CQDMEM2 = CQDMEM1     165  ELEMENTS
  GROUP     2    CROD    = CROD        451  ELEMENTS
  GROUP     3    CSHEAR  = CSHEAR       96  ELEMENTS
  GROUP     4    CTRMEM  = CTRMEM       96  ELEMENTS
 ? READ

END
 MODULE READ      ENDED, TIME =00:14:40  DELTA =   41.01
 ? CADS

SET
 ? SET

E1 ALL
 ? SET
```

This is a NASTRAN model of a wing structure. This example also uses the Tektronix 4014 terminal and the NASTRAN communications mode. There is no POST data base and no existing GEOMETRY data base. A new geometry data base, called GEOMNAST2.DAT, will be created. The READ module is entered and the BEGIN NASTRAN command is used to read in the NASTRAN bulk data deck named: NASTIN2.DAT. The model groups are printed out; the READ module is ended; and the SET module is started. The element set E1, composed of all the model's elements, is defined using the E1 ALL command. The E1 set was then sent to the DISPLAY module.

```
 ? DISPLAY
ROTATE  Y 45.0 Z 45.0
 ? DISPLAY
PLOT
```

The element set was then rotated about the Y and Z display axes by 45.0 degrees. The PLOT command is then used to display the element set.



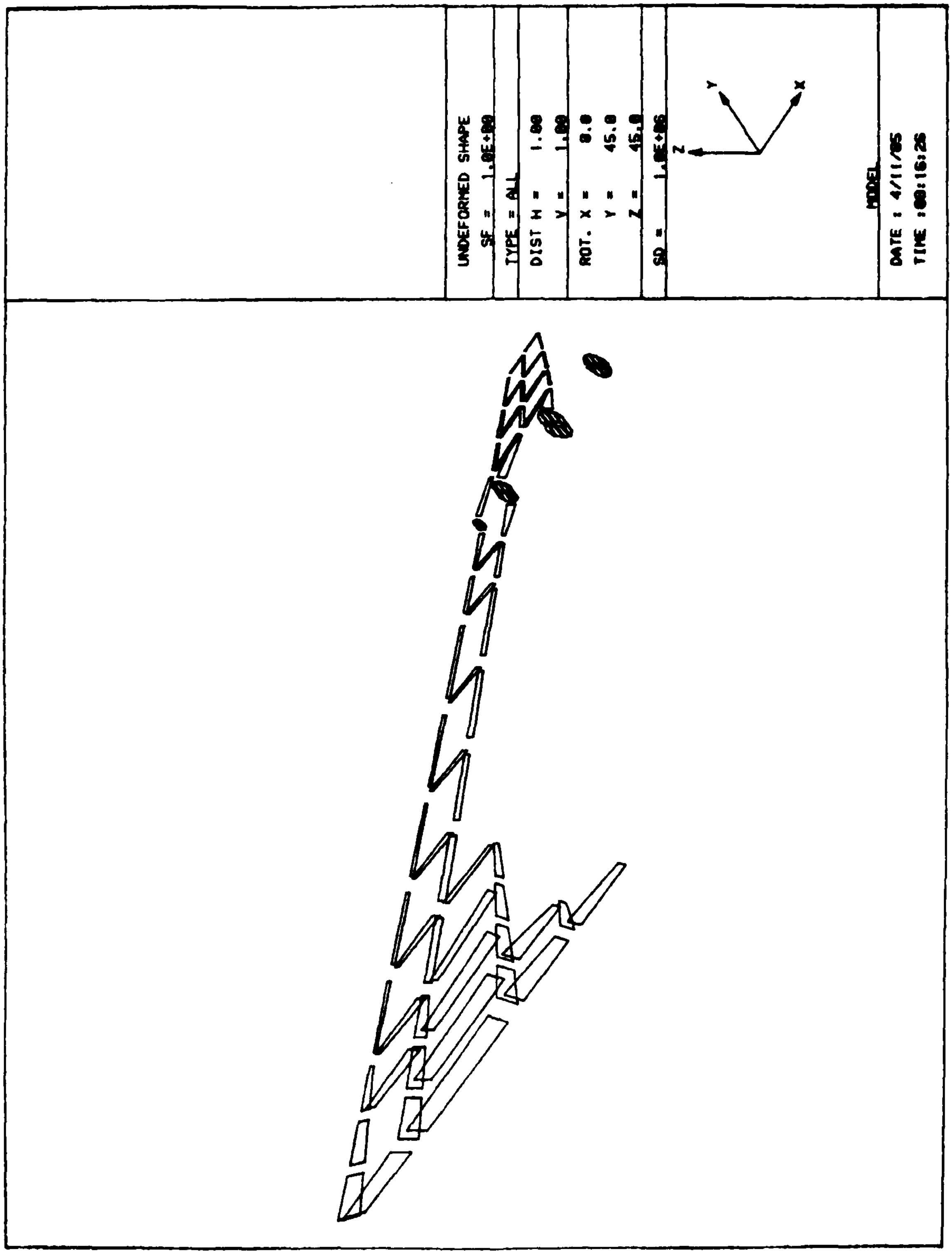
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:15:50

```
? DISPLAY
DEFINE CSHEAR
```

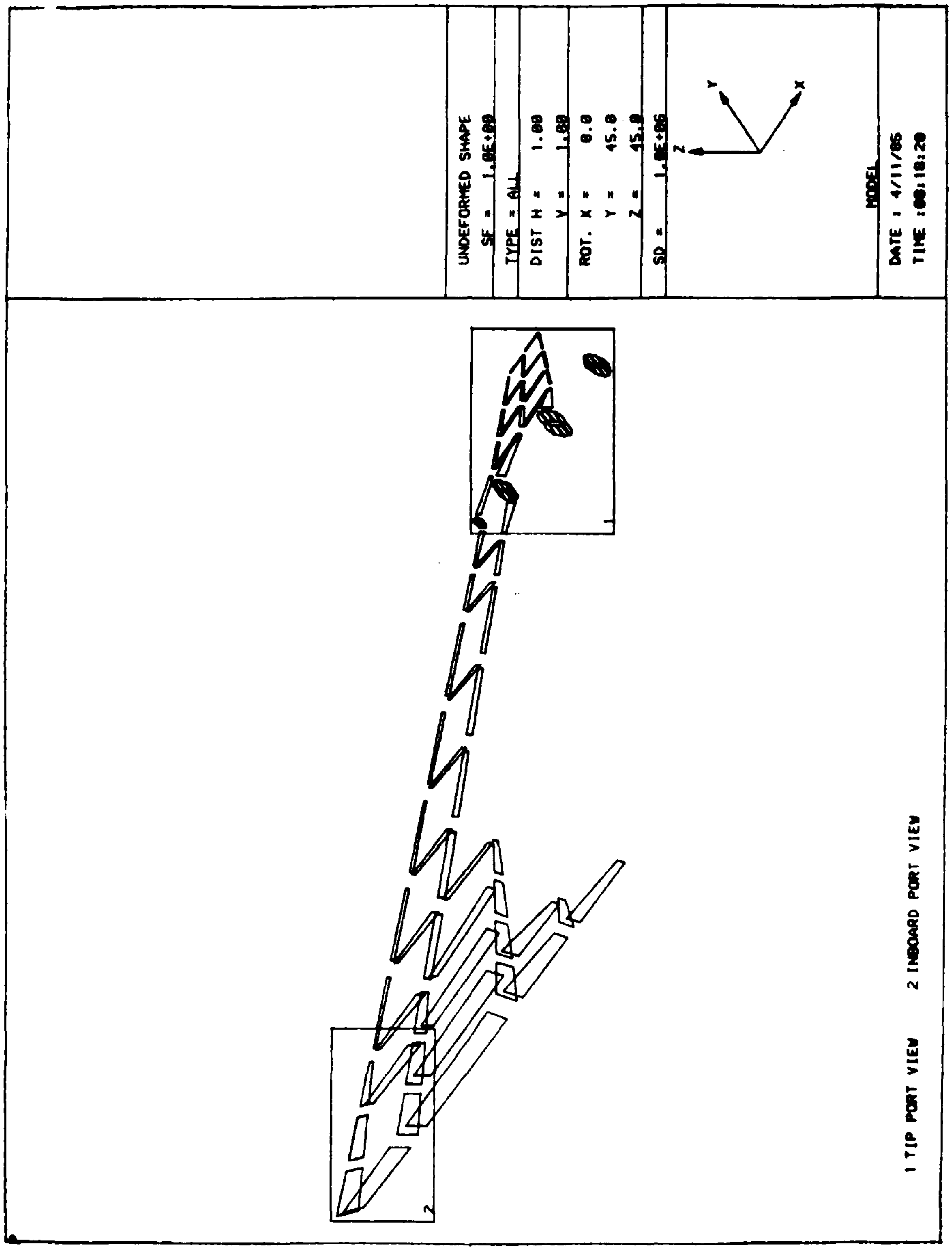
A new element set composed of the CSHEAR elements in the model was defined using the DEFINE CSHEAR command. The screen is then cleared before additional commands are issued.

```
? DISPLAY
PLOT BREAK
```

The PLOT BREAK command is now used to display this new element set of CSHEARs as individual elements.

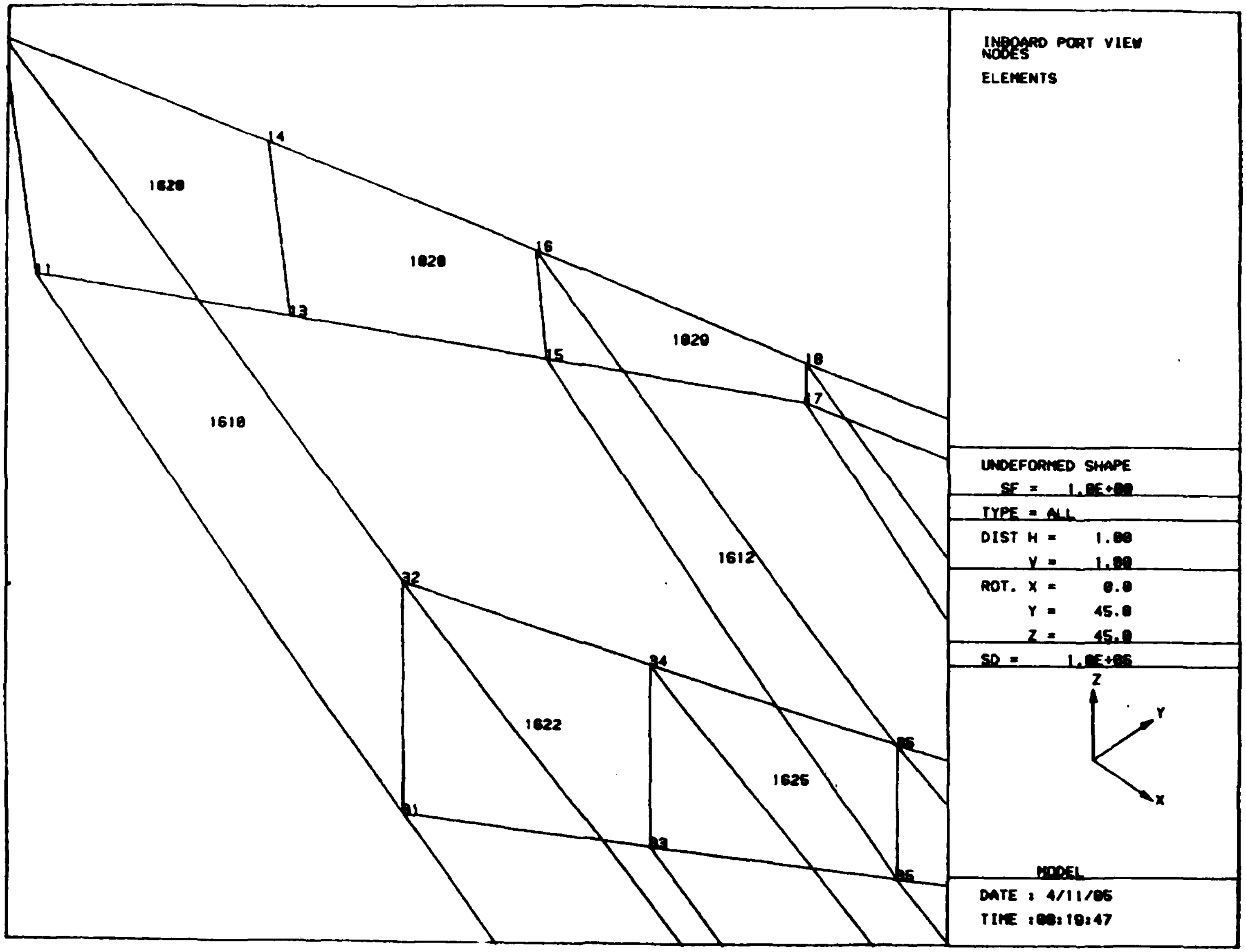

UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:16:26

This set was then plotted and the PORT (P) processing available at the end of a plot was illustrated. Instead of the usual blank character a P was entered after the plot was completed. This started the crosshairs which were then used to define two ports. The ports are defined by moving the crosshairs to the lower left corner of a box where an integer number (1-9) is pressed; then the crosshairs are moved to the upper right corner of the box and an integer number is again pressed; finally a title is entered at the bottom of the screen. This process was repeated for the second port and the return (R) character was entered to return to the DISPLAY module.


UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Y
X
Z
MODEL
DATE : 4/11/86
TIME :08:18:20
1 TIP PORT VIEW
2 INBOARD PORT VIEW

```
? DISPLAY
PLOT PORT 2 ELEMENT NODE
```

The second port will now be displayed. A port takes the elements in the solid box defined by the previous port processing and expands them to fill the entire display area of the terminal. The PLOT command with the PORT 2 keyword specifies that the port box numbered 2 will be expanded. The ELEMENT and NODE keywords turn on the node and element numbering for the display as shown on the following page. Note the port title at the top of the margin legend box.



INBOARD PORT VIEW
NODES
ELEMENTS
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME :08:19:47
1020
1020
1020
1610
1612
1622
1625
14
16
18
13
15
17
32
34
36
31
33
35

```
 ? DISPLAY
EDIT
 ? EDIT
BEGIN CASE
 ? EDITCA
LIST NASTRAN 1 TO 20
     NASTRAN CASE CONTROL DECK
   1  ID WING MODEL
   2  APP DISP
   3  SOL 3,0
   4  TIME,60
   5  CEND
   6  DISP(PRINT,PUNCH) = ALL
   7  SPC = 1
   8  METHOD = 10
   9  BEGIN BULK
  10  GRDSET                                                        456
  11  EIGR          10     INV     0.0   200.0      10       3           1.0E-03
+ABC
  12  +ABC         MAX
  13  SPC1           1     123       1      11      12      31      32      51
+BCD
  14  +BCD          52      67      68      75      78    8001
 ? EDITCA
REPLACE NASTRAN 3
 ENTER A CASE CONTROL CARD TO REPLACE AN OLD ONE
 ? REPLACE
SOL 1,0
 ? EDITCA
LIST NAS 1 TO 5
     NASTRAN CASE CONTROL DECK
   1  ID WING MODEL
   2  APP DISP
   3  SOL 1,0
   4  TIME,60
   5  CEND
 ? EDITCA
END
 MODULE EDITCA   ENDED, TIME =08:41:51  DELTA =  106.09
 ? EDIT
```

This is a CASE EDIT example. The LIST NASTRAN 1 TO 20 command lists the first 20 non-geometry cards stored on the data base. These are typically NASTRAN executive, case, and comment cards. Note there are only 14 such cards in this model. The third card is replaced using the REPLACE NASTRAN 3 command followed by the new card which is replacing the existing third card. LIST NAS 1 TO 5 is used to verify the change and EDIT CASE is ended. Note the NAS abbreviation of NASTRAN.

```
SAVE
 ? EDIT

END
 MODULE EDIT      ENDED, TIME =08:21:58  DELTA =   10.13
 ? CADS

END
 MODULE CADS      ENDED, TIME =08:22:04  DELTA =    5.37
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
N
FORTRAN STOP
$
```

The changes are permanently saved using the SAVE command of the editor and the editor is ended. SAVE saves the changes to the existing geometry data base; thus deleting the old data. The COPY command of the editor may be used to copy the new data to a new geometry data base file. CADS is then ended.

12.6 NASTRAN WING BOX TEST CASE

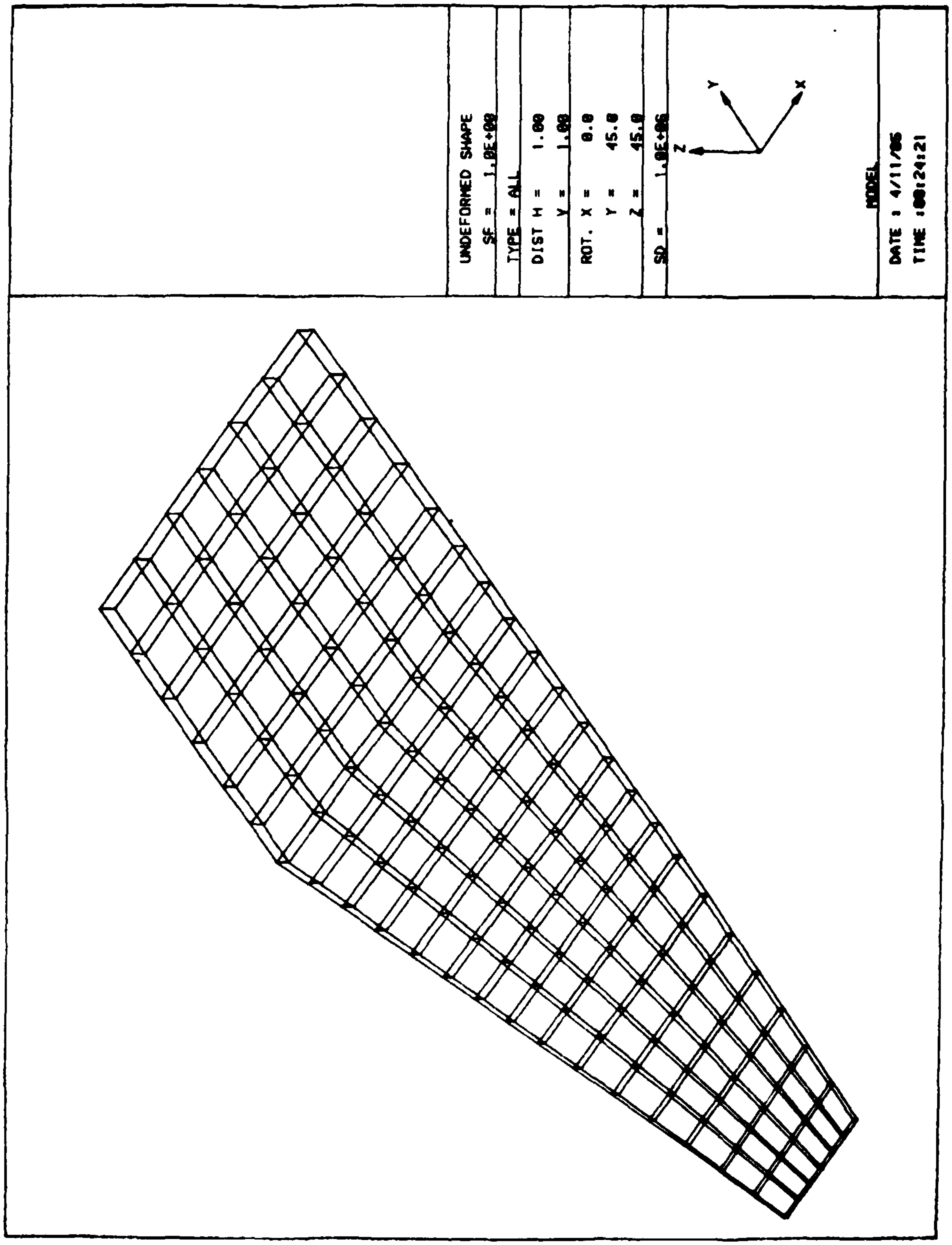
This case is an anisotropic wing box model with two simple loading conditions and is used to illustrate the use of the DISPLAY module commands dealing with analysis program output data. In particular contour, value, deformed, and X-Y graph examples of NASTRAN analysis outputs are shown.

```
RUN CADS
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NASTRAN
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
Y
 ENTER POST DATA BASE FILE NAME FOR CADS PLOTTING OR END TO SKIP
POSTNAST3.DAT
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
Y
 ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP
GEOMNAST3.DAT
 ? CADS

SET
 ? SET

LIST GROUP
  GROUP    5    CONROD  = CONROD     154  ELEMENTS
  GROUP   10    CONROD  = CONROD      64  ELEMENTS
  GROUP   12    CONROD  = CONROD      64  ELEMENTS
  GROUP   21    CONROD  = CONROD      10  ELEMENTS
  GROUP   23    CONROD  = CONROD      10  ELEMENTS
  GROUP   25    CONROD  = CONROD      10  ELEMENTS
  GROUP    1    CQDMEM1 = CQDMEM1    126  ELEMENTS
  GROUP    2    CQDMEM1 = CQDMEM1    126  ELEMENTS
  GROUP    3    CSHEAR  = CSHEAR     147  ELEMENTS
  GROUP    4    CSHEAR  = CSHEAR     132  ELEMENTS
  GROUP   11    CSHEAR  = CSHEAR      21  ELEMENTS
  GROUP   13    CSHEAR  = CSHEAR      21  ELEMENTS
  GROUP   22    CSHEAR  = CSHEAR       6  ELEMENTS
  GROUP   24    CSHEAR  = CSHEAR       6  ELEMENTS
  GROUP   26    CSHEAR  = CSHEAR       6  ELEMENTS
 ? SET

E1 ALL
 ? SET

PLOT E1
```

This is an anisotropic material wing box structure model with a post data base (POSTNAST3.DAT) and a geometry data base (GEOMNAST3.DAT). The geometry data base was generated in a previous execution of CADS and the VAX file name given to the data base at that time is now specified as the existing geometry data base name. The data base files are attached and the SET module is begun. The model's groups are then listed by LI GROUP where LI is the abbreviation for LIST. Set E1 is defined as all of the model's elements and is then sent to the DISPLAY module through the E1 ALL and PLOT E1 commands.

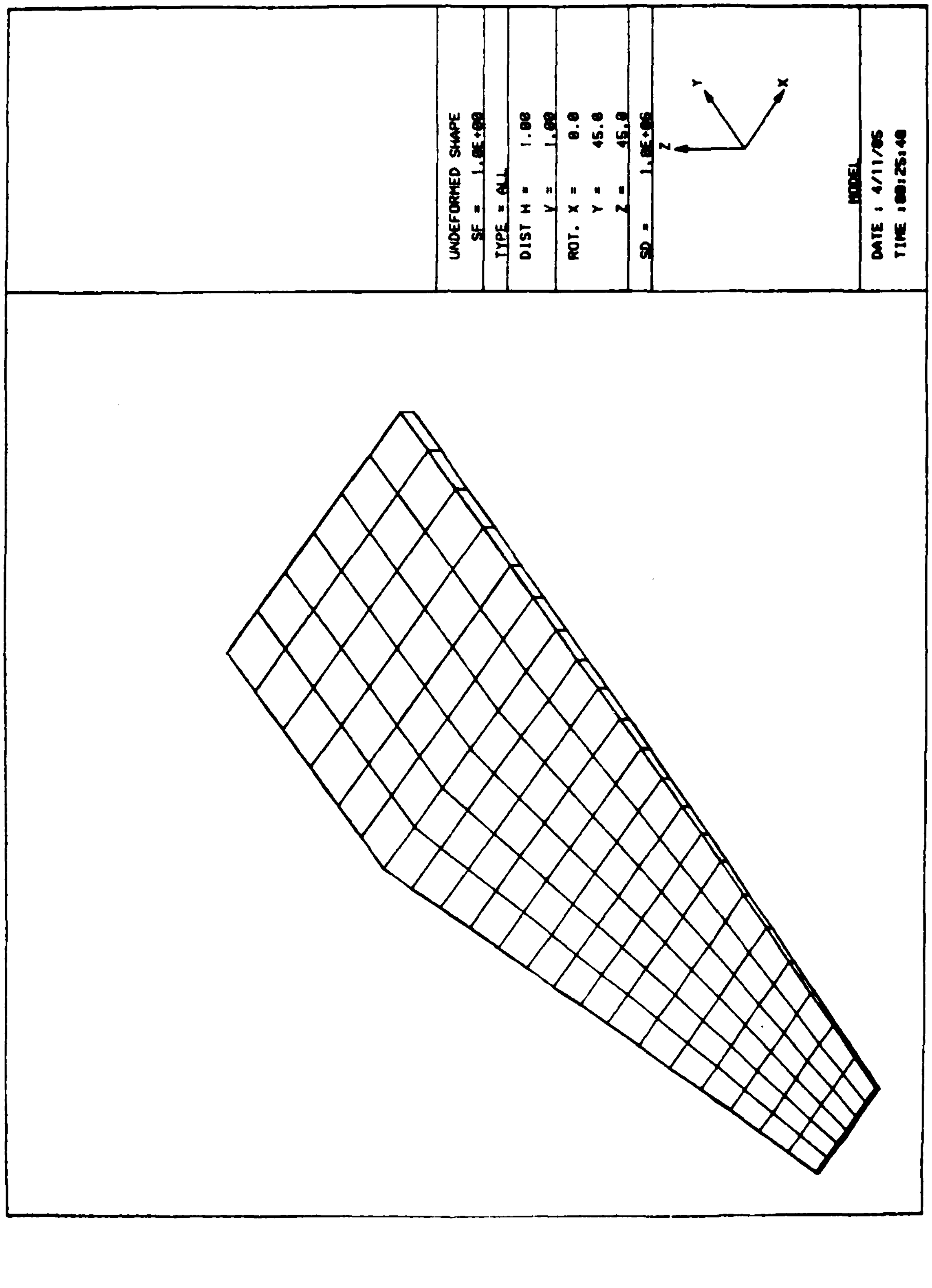
```
? DISPLAY
ROTATE X 0  Y 45  Z 45
? DISPLAY
PLOT
```

The elements are rotated about the X, Y, and Z display axes using the ROTATE command and the model is then plotted using PLOT.


UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME :00:24:21

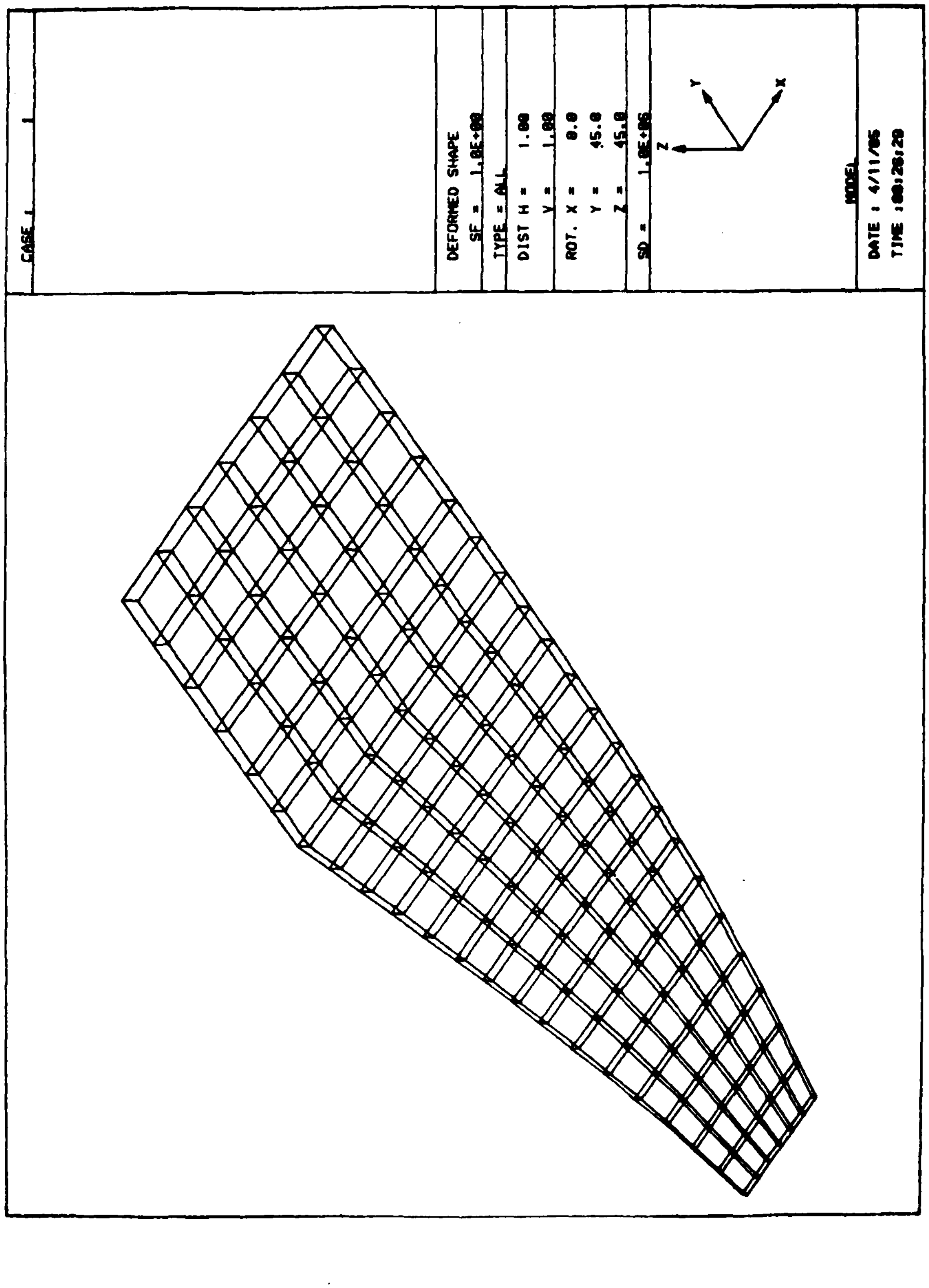
```
 ? DISPLAY
ROTATE X 0  Y 45  Z 45
 ? DISPLAY
PLOT HIDE
```

A hidden line view is then made using the HIDE keyword on the PLOT command. The result is on the next page.


UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:25:40

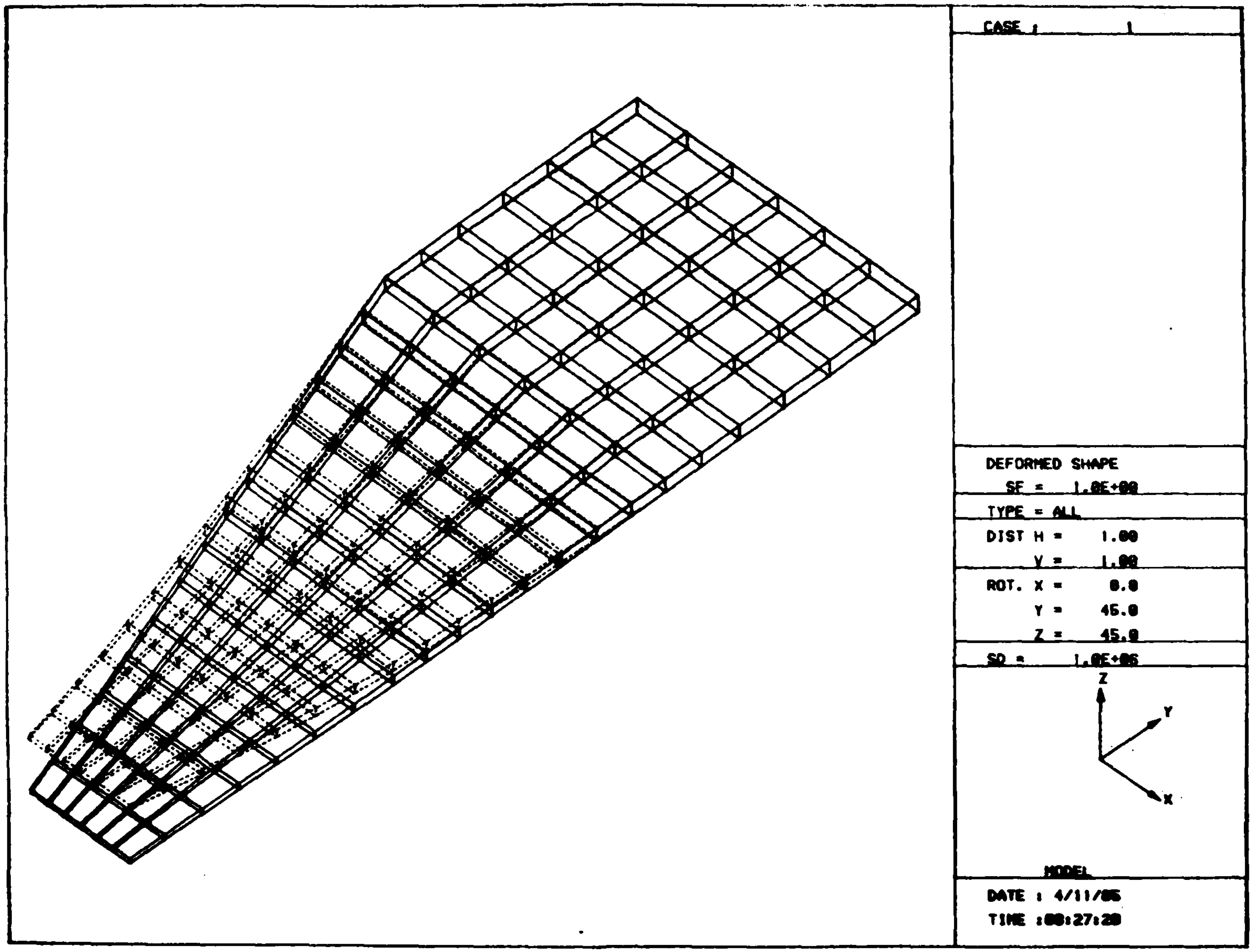
```
 ? DISPLAY
CASE 1
 ? DISPLAY
MODE DISPLACE
 ? DISPLAY
DEFORM
 ? DISPLAY
PLOT
```

The CASE 1 command is used to specify the analysis results with the NASTRAN subcase numbered 1 for display. The MODE DISPLACE command specifies that the displacement data be used while the DEFORM command is used to turn on the deformed shape plotting. Deformed shapes will now be plotted any time a PLOT command is issued until a new set is specified or the NODEFORM command is given. Note the slightly curved shape at the tip of the model.


CASE 1
DEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME :08:26:29

```
? DISPLAY
PLOT BOTH
```

The BOTH keyword of the PLOT command will plot the deformed shape on top of the undeformed shape. The deformed shape is in dashed lines while the undeformed shape is in the solid lines.

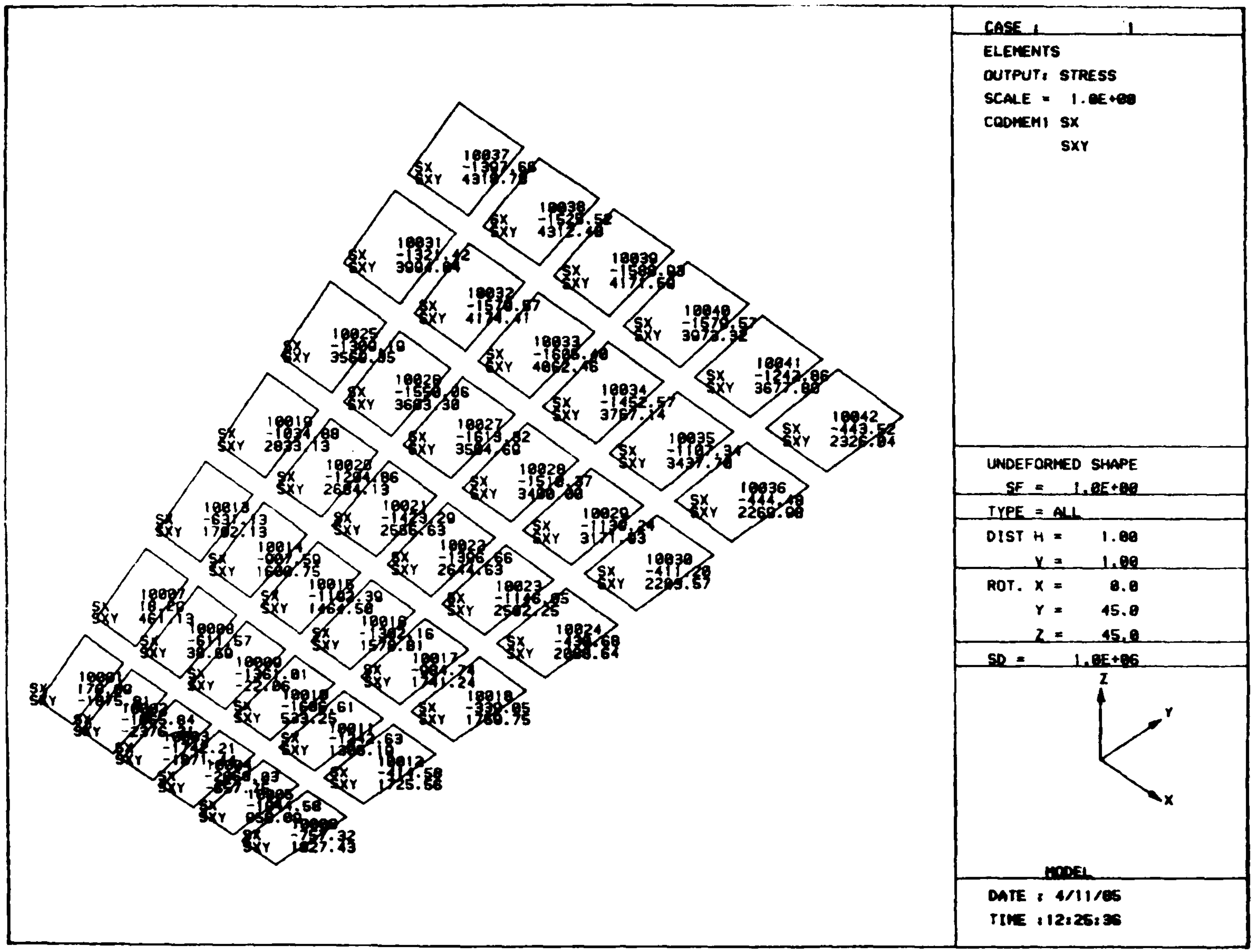

CASE : 1
DEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:27:20

```
 ? DISPLAY

LIST GROUP
  GROUP    5    CONROD  = CONROD     154  ELEMENTS
  GROUP   10    CONROD  = CONROD      64  ELEMENTS
  GROUP   12    CONROD  = CONROD      64  ELEMENTS
  GROUP   21    CONROD  = CONROD      10  ELEMENTS
  GROUP   23    CONROD  = CONROD      10  ELEMENTS
  GROUP   25    CONROD  = CONROD      10  ELEMENTS
  GROUP    1    CQDMEM1 = CQDMEM1    126  ELEMENTS
  GROUP    2    CQDMEM1 = CQDMEM1    126  ELEMENTS
  GROUP    3    CSHEAR  = CSHEAR     147  ELEMENTS
  GROUP    4    CSHEAR  = CSHEAR     132  ELEMENTS
  GROUP   11    CSHEAR  = CSHEAR      21  ELEMENTS
  GROUP   13    CSHEAR  = CSHEAR      21  ELEMENTS
  GROUP   22    CSHEAR  = CSHEAR       6  ELEMENTS
  GROUP   24    CSHEAR  = CSHEAR       6  ELEMENTS
  GROUP   26    CSHEAR  = CSHEAR       6  ELEMENTS
 ? DISPLAY

NODEFORM
 ? DISPLAY

CASE 1
 ? DISPLAY

BEGIN ATTRIB
 ? ATTRIB

PROGRAM NASTRAN
 ? ATTRIB

MODE STRESS
 ? ATTRIB

CQDMEM1 SX SXY
 ? ATTRIB

END
 MODULE ATTRIB   ENDED, TIME =00:20:17  DELTA =  320.57
 ? DISPLAY

DEFINE GROUP 1 EL 1 TO 42
```

The LIST command is given to review the model's groups and NODEFORM is specified to turn off the deformed plotting. The ATTRIBUTE module is entered; the NASTRAN analysis program is defined and the STRESS mode is turned on. The CQDMEM1 element is called out with the SX and SXY stress components. The module is ended and a new element set is defined for plotting as given by the DEFINE command.

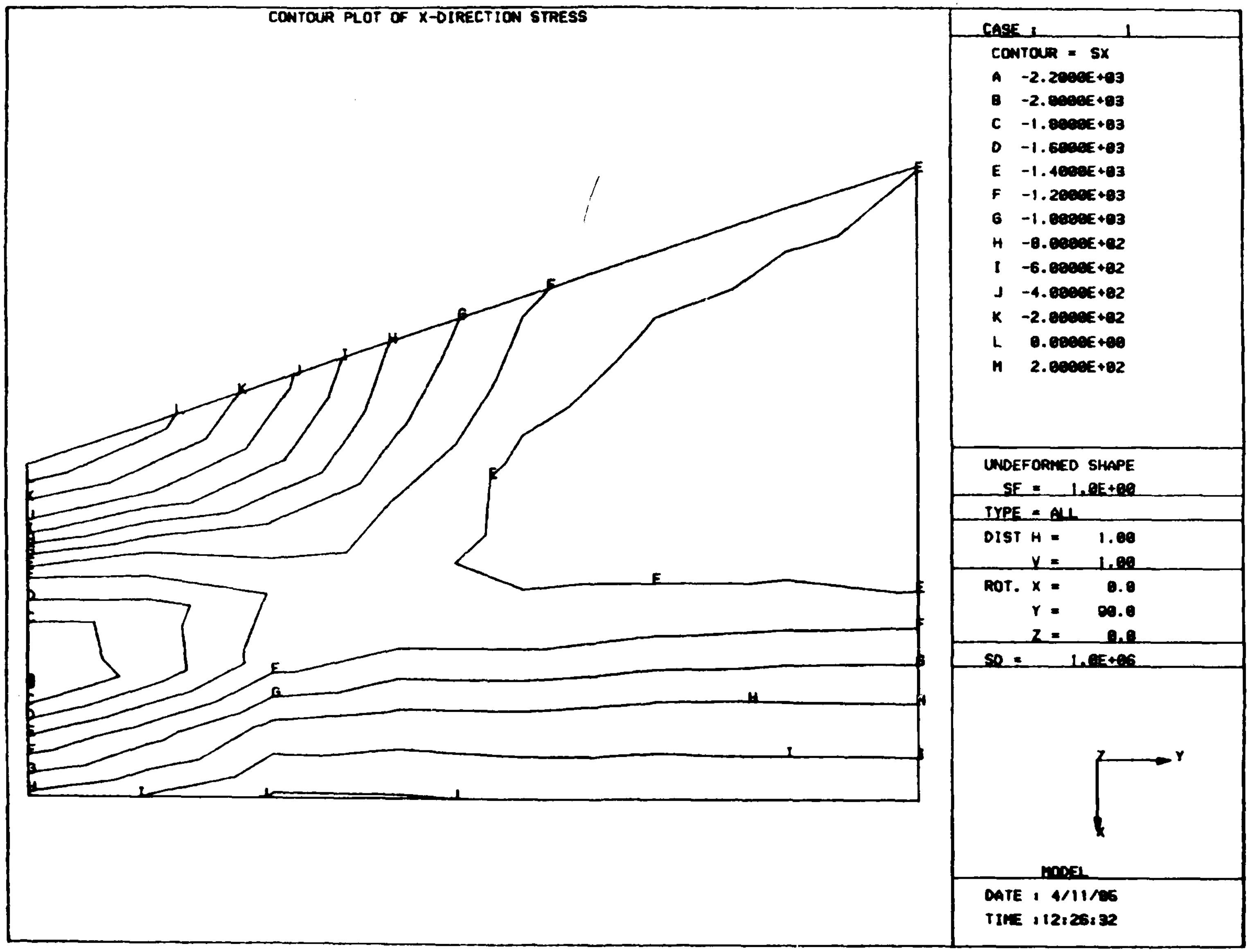
```
7 DISPLAY
PLOT ELEMENT BREAK STRESS
```

This set is now plotted with the element numbers, the break option and the element stresses displayed. Note that the stress component name and value are displayed on each element. The output information component names are also summarized in the margin.


CASE : 1
ELEMENTS
OUTPUT: STRESS
SCALE = 1.0E+00
CQDMEM1 SX
SXY
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :12:25:36

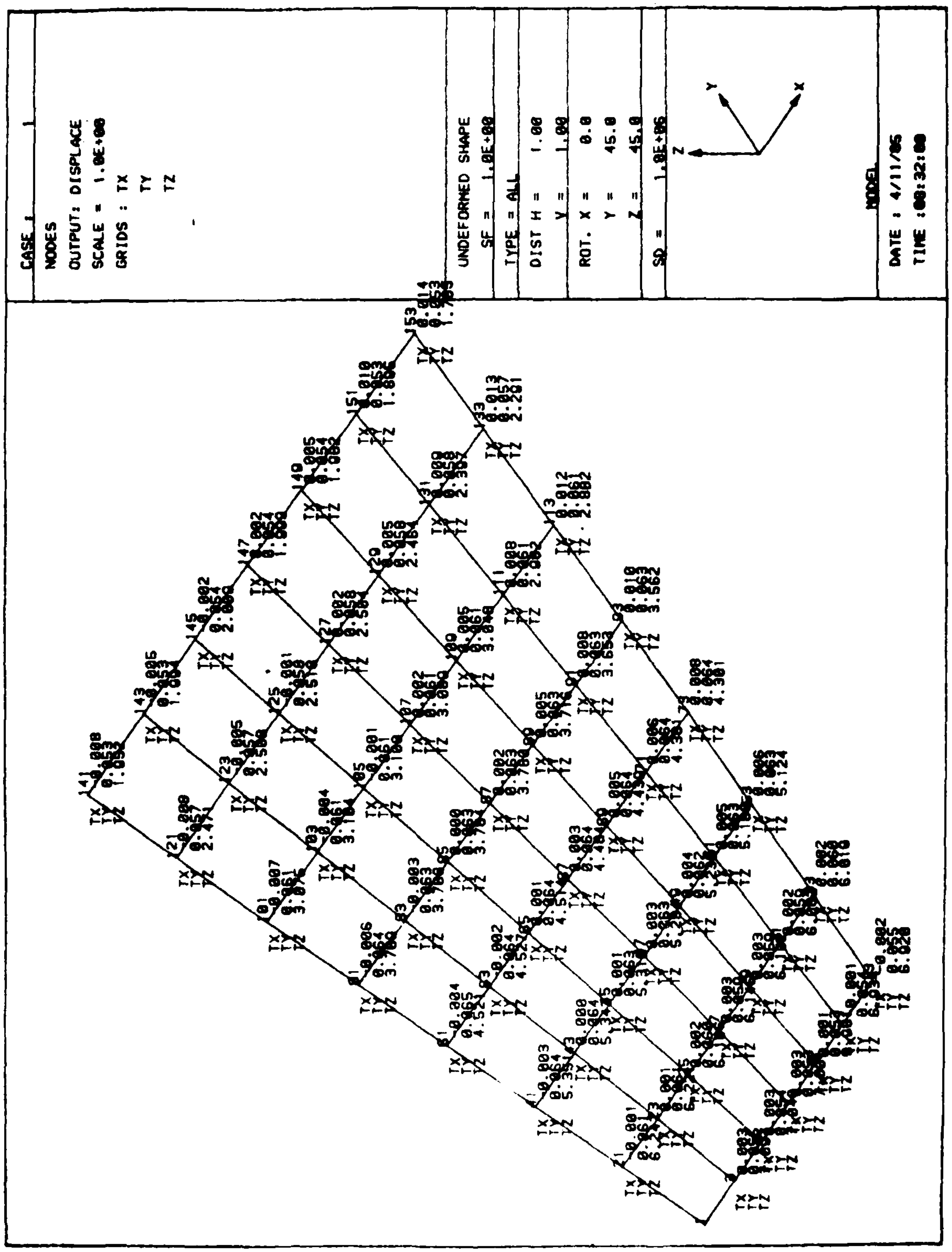
```
? DISPLAY

ROTATE Y 90 Z 0
 ? DISPLAY

TITLE
CONTOUR PLOT OF X-DIRECTION STRESS
 ? DISPLAY

PLOT CONTOUR STRESS
```

The display is rotated to Y=90.0 and Z=0.0 degrees. A plot title is specified following the TITLE command. This title remains on until specifically turned off or changed. A contour plot of the SX stresses is then requested using the CONTOUR STRESS keywords on the PLOT command. Note the automatic level definition for the contour values. The SX values are contoured since the contour keyword uses the first component type defined on the element in the ATTRIBUTE module to define the value to be contoured.


CONTOUR PLOT OF X-DIRECTION STRESS
CASE : 1
CONTOUR = SX
A -2.2000E+03
B -2.0000E+03
C -1.8000E+03
D -1.6000E+03
E -1.4000E+03
F -1.2000E+03
G -1.0000E+03
H -8.0000E+02
I -6.0000E+02
J -4.0000E+02
K -2.0000E+02
L 0.0000E+00
M 2.0000E+02
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 90.0
Z = 0.0
SD = 1.0E+06
MODEL
DATE : 4/11/86
TIME :12:26:32

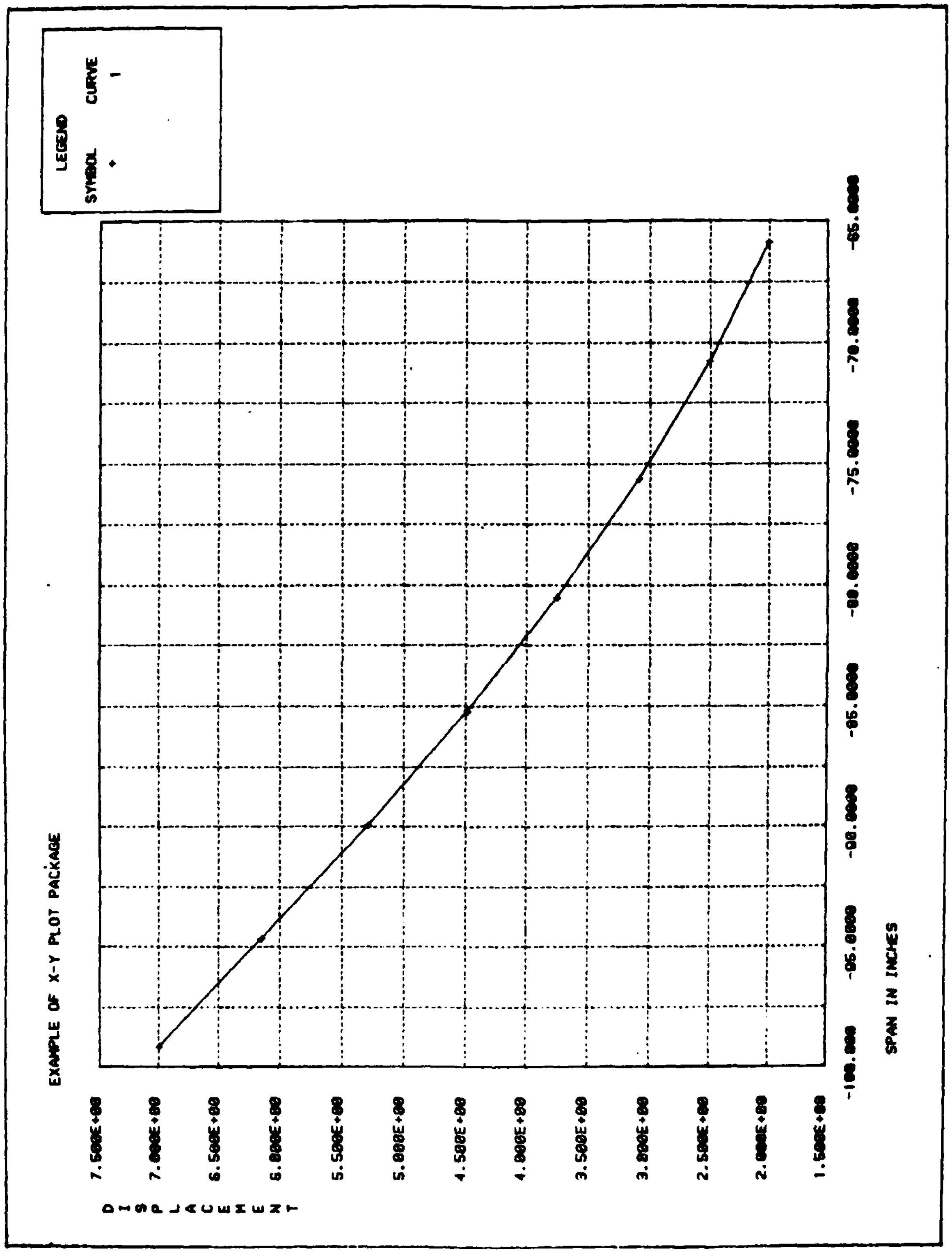
```
 ? DISPLAY
NOTITLE
 ? DISPLAY
BEGIN ATTRIB
 ? ATTRIB
CLEAR ALL
 ? ATTRIB
PROGRAM NASTRAN
 ? ATTRIB
MODE DISPLACE
 ? ATTRIB
NODE TX TY TZ
 ? ATTRIB
END
 MODULE ATTRIB   ENDED, TIME =08:31:34  DELTA =  107.83
 ? DISPLAY
ROTATE Y 45  Z 45
 ? DISPLAY
PLOT NODE DISPLACE
```

The NOTITLE command turns off the previous title. The ATTRIBUTE module is entered, and the NASTRAN DISPLACEMENT mode is specified. The NODE type is defined with component names TX, TY, and TZ before the module is ended. The display is rotated to Y and Z of 45 degrees and then plotted with the node numbers and displacement values.


CASE 1
NODES
OUTPUT: DISPLACE
SCALE = 1.0E+00
GRIDS : TX
TY
TZ
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
MODEL

```
 ? DISPLAY
GRAPH
 ? GRAPH
CURVE 1
 ? GRAPH
CASE 1
 ? GRAPH
TITLE
EXAMPLE OF X-Y PLOT PACKAGE
 ? GRAPH
YTITLE
DISPLACEMENT
 ? GRAPH
XTITLE
SPAN IN INCHES
 ? GRAPH
XVALUE 1 AXIS Y NODE 7 TO 147 BY 20
 ? GRAPH
YVALUE 1 DISP TZ NODE 7 TO 147 BY 20
 ? GRAPH
EXECUTE
```

The X-Y graph processing is started with GRAPH. One curve will be drawn using data from load case 1 as specified by the CURVE 1 and CASE 1 commands. The X-Y graph title, X-axis title, and Y-axis title are defined using the TITLE, XTITLE, and YTITLE commands, respectively. Finally the X-values and Y-values are defined and the plot is executed. The X's are the Y coordinates for nodes 7 to 147 by 20. The Y's are the Z displacements for those nodes. Following the display of the X-Y graph two END commands are given to get back to the ? CADS Executive Monitor for starting a new model.



EXAMPLE OF X-Y PLOT PACKAGE
LEGEND
SYMBOL
CURVE
1
DISPLACEMENT
7.500E+00
7.000E+00
6.500E+00
6.000E+00
5.500E+00
5.000E+00
4.500E+00
4.000E+00
3.500E+00
3.000E+00
2.500E+00
2.000E+00
1.500E+00
-100.000
-95.0000
-90.0000
-85.0000
-80.0000
-75.0000
-70.0000
-65.0000
SPAN IN INCHES

12.7 NATURAL ALL TEST

This case contains small samples of all the various element, grid point, load and material commands. It is used to provide a simple sample of many of the NATURAL generation input commands. The second execution of this model shows how to attach an existing geometry data base and use the editting functions to modify, save, and copy the geometry data base for future use.

```
 ? CADS

END
 MODULE CADS     ENDED, TIME =08:37:32  DELTA =    2.50
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
Y
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ...., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NATURAL
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
N
 ENTER THE TITLE TO MODEL HEADER
TEST EXAMPLE FOR ALL ELEMENTS
 ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP
GEOMNATU1.DAT
 ? CADS

READ
 ? READ

BEGIN NATURAL INPUT 20
 ENTER NATURAL INPUT FILE NAME NOW OR END TO RETURN
NATUIN1.DAT
```

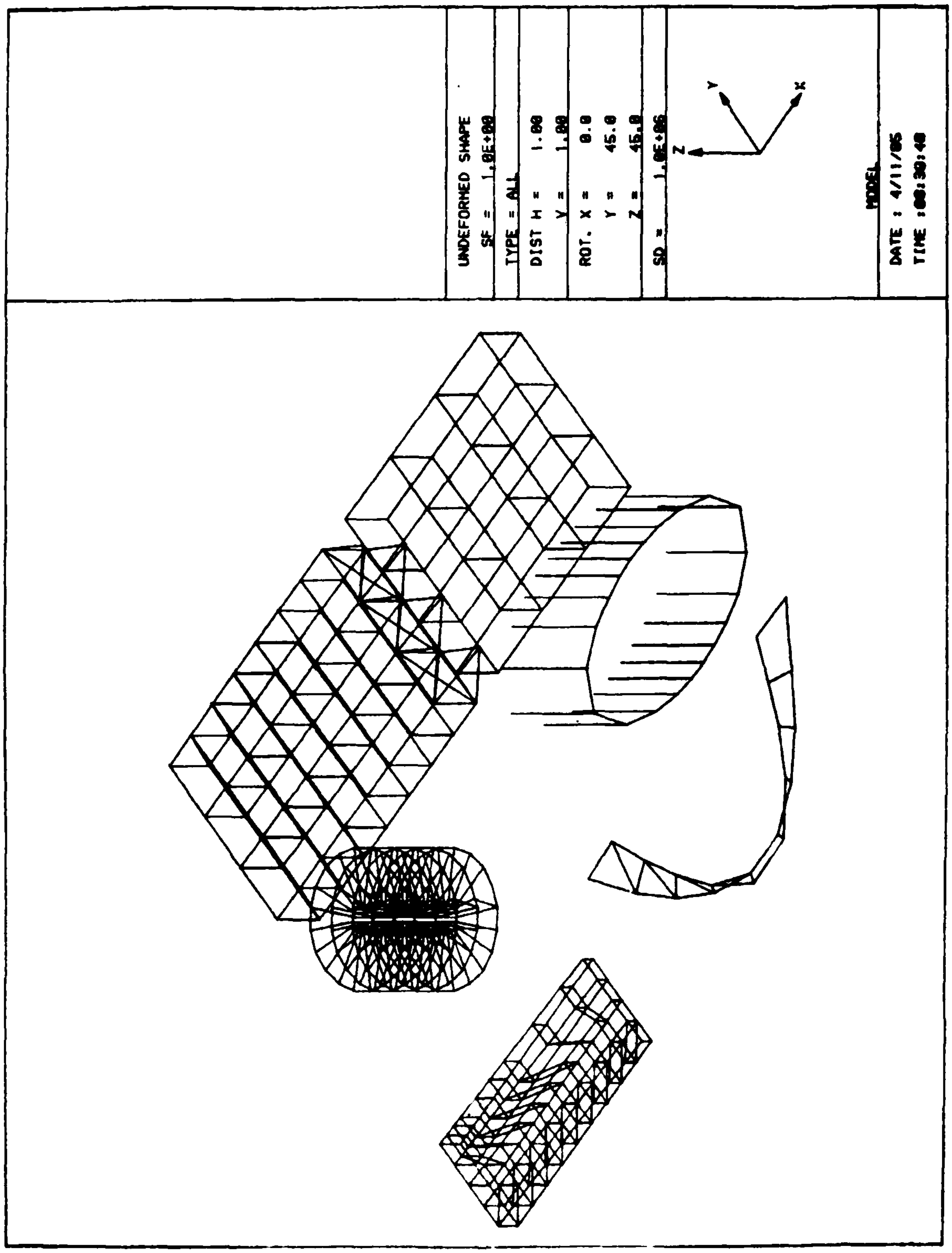
The anisotropic wing box model is ended and another model is begun using the END and Y inputs. Thus the processing of a nautral generation model which contains examples of the different generation commands is started. The communications mode is NATURAL so the element names used will be the natural generation names. A new geometry data base is being created and it is called GEOMNATU1.DAT. The INPUT 20 keyword is used with the BEGIN NATURAL command to specify the unit for the input steering file of generation commands. By default the NATURAL processor expects input commands from the terminal. NATUIN1.DAT contains these sample commands.

```
MODULE DIRECT   ENDED, TIME =00:30:27  DELTA =  53.540
MODULE SHAPE    ENDED, TIME =00:30:29  DELTA =  55.340
MODULE DIRECT   ENDED, TIME =00:30:30  DELTA =  56.300
MODULE FREEDOM  ENDED, TIME =00:30:31  DELTA =  50.100
MODULE LOAD     ENDED, TIME =00:30:32  DELTA =  50.600
MODULE NODES    ENDED, TIME =00:30:32  DELTA =   50.60
MODULE ELEMENT  ENDED, TIME =00:30:34  DELTA =   2.350
MODULE DIRECT   ENDED, TIME =00:30:42  DELTA =   0.600
MODULE ANIS     ENDED, TIME =00:30:42  DELTA =   10.40
MODULE PROPERTY ENDED, TIME =00:30:43  DELTA =   0.400
MODULE NATURAL  ENDED, TIME =00:30:43  DELTA =    0.57
    26  ACCEPTABLE GROUPS PROCESSED

 GROUP    1    CONROD  = R2          16  ELEMENTS
 GROUP    2    CONROD  = R2          16  ELEMENTS
 GROUP    3    CONROD  = R2          50  ELEMENTS
 GROUP    4    CBAR    = B2          36  ELEMENTS
 GROUP    5    CSHEAR  = QS4         16  ELEMENTS
 GROUP    6    CTWIST  = QT4         10  ELEMENTS
 GROUP    7    CTRMEM  = TM           0  ELEMENTS
 GROUP    8    CTRIA3  = TB3          0  ELEMENTS
 GROUP    9    CQDMEM1 = QM1          0  ELEMENTS
 GROUP   10    CQUAD4  = QB4         10  ELEMENTS
 GROUP   11    CTRIA1  = TB1          0  ELEMENTS
 GROUP   12    CTRIA2  = TB2          0  ELEMENTS
 GROUP   13    CQUAD1  = QB1          0  ELEMENTS
 GROUP   14    CQUAD2  = QB2         10  ELEMENTS
 GROUP   21    CQDMEM1 = QM1          5  ELEMENTS
 GROUP   22    CTRMEM  = TM           0  ELEMENTS
 GROUP   31    CQM8    = QM8         05  ELEMENTS
 GROUP   32    CTRIM6  = TM6         10  ELEMENTS
 GROUP   33    CONROD  = R2          00  ELEMENTS
 GROUP   34    CONROD  = R2         100  ELEMENTS
 GROUP   41    CELAS1  = AS          20  ELEMENTS
 GROUP   43    CONROD  = R2          00  ELEMENTS
 GROUP   51    CWEDGE  = SO6         10  ELEMENTS
 GROUP   52    CIHEX1  = SO8         12  ELEMENTS
 GROUP   53    CIHEX2  = SO20         9  ELEMENTS
 GROUP   54    CTETRA  = SO4          6  ELEMENTS
 ? READ

END
 MODULE READ    ENDED, TIME =00:30:46  DELTA =    2.00
 ? CADS

SET
 ? SET

E1 ALL
 ? SET

PLOT E1
```

The various generation modules are automatically entered and exited as the geometry data is processed. The model's groups are summarized at the end; the SET module is entered and all of the elements are placed in the set E1. This set is passed to the DISPLAY module using the PLOT E1 command.

```
? DISPLAY
ROTATE Y 45  Z 45
 ? DISPLAY
PLOT
```

The elements are rotated and then displayed using the standard ROTATE and PLOT commands. Since this is meant to be an example model, the plots do not represent any real structure which is to be analyzed.

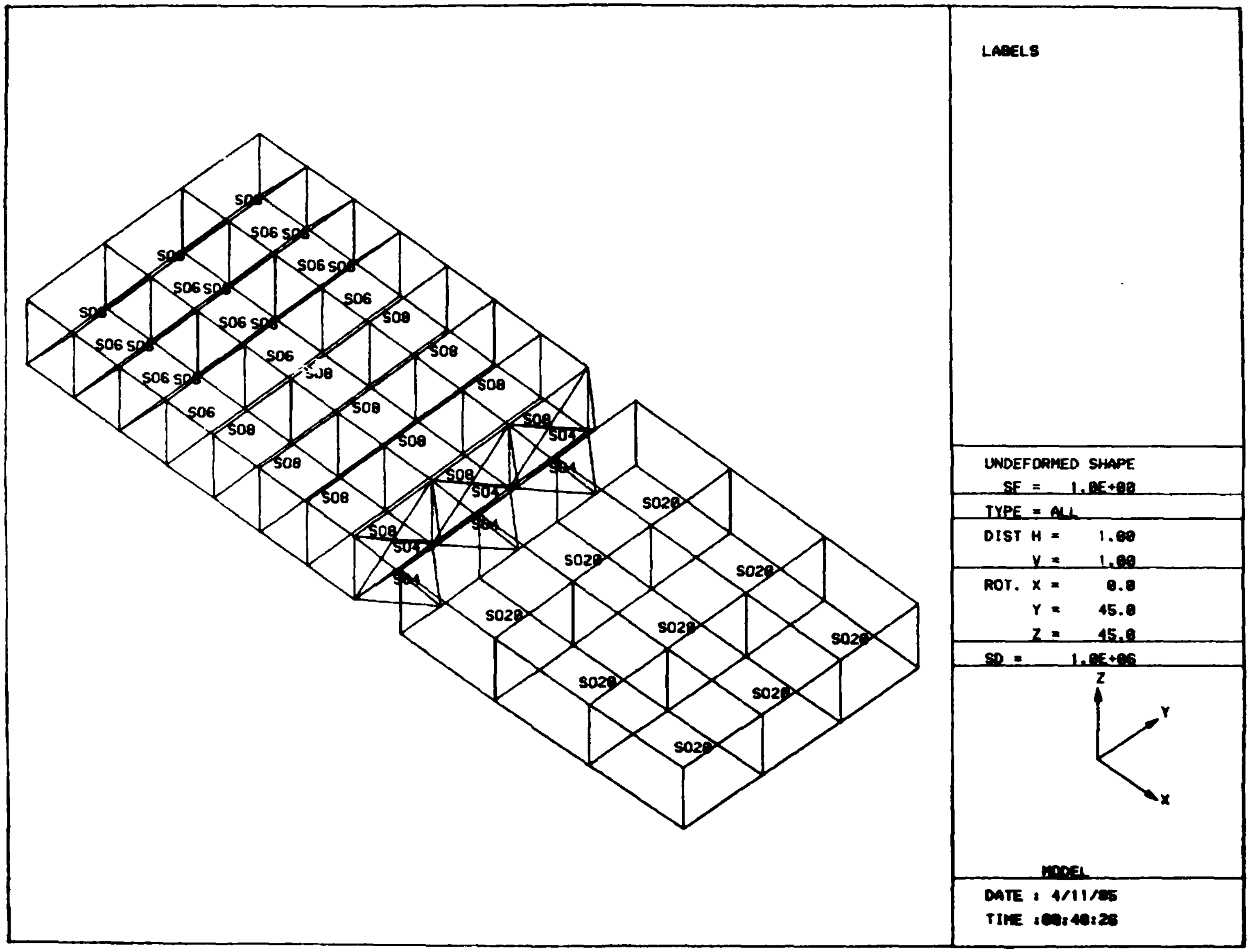

UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:30:40

```
? DISPLAY
DEFINE GROUP 51 TO 54
```

The DEFINE command is now used to get groups 51 to 54 which make up the large box displayed in the upper right quadrant of the previous figure.

```
? DISPLAY
PLOT LABEL
```

The LABEL keyword is now used to label the elements in the next display with their element names. Note the element names are in terms of the NATURAL communications mode since that was the mode specified at the program's initialization.


LABELS
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:40:26

```
? DISPLAY
DEFINE GROUP 7 8 9 10
```

Another new element set is defined consisting of groups 7, 8, 9, and 10. These are cover elements on the small box structure in the lower left quadrant of the earlier display containing all of the elements.

```
? DISPLAY

ROTATE Y 90  Z 90
 ? DISPLAY

COLOR
 ? DISPLAY

PLOT NODE ELEMENT BREAK
```

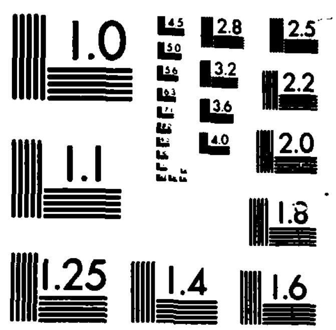
The screen was cleared. The new elements are now rotated to Y and Z of 90.0 degrees. The COLOR command is issued to turn on the color processing which will now remain on until specifically turned off. Each element type will now be displayed using a different color or pattern. On a 4014 terminal the DI-3000 graphics package used by CADS will make each color a different pattern. The node and element numbers will be shown on a shrunken element display using the PLOT command with the NODE, ELEMENT, and BREAK keywords.

AD-A163 258 CADS - A COMPUTER AIDED DESIGN SYSTEM VOLUME 2 USER'S GUIDE(U) ROCKWELL INTERNATIONAL LOS ANGELES CA NORTH AMERICAN AIRCRAFT.. M C LESS ET AL. AUG 85 3/3

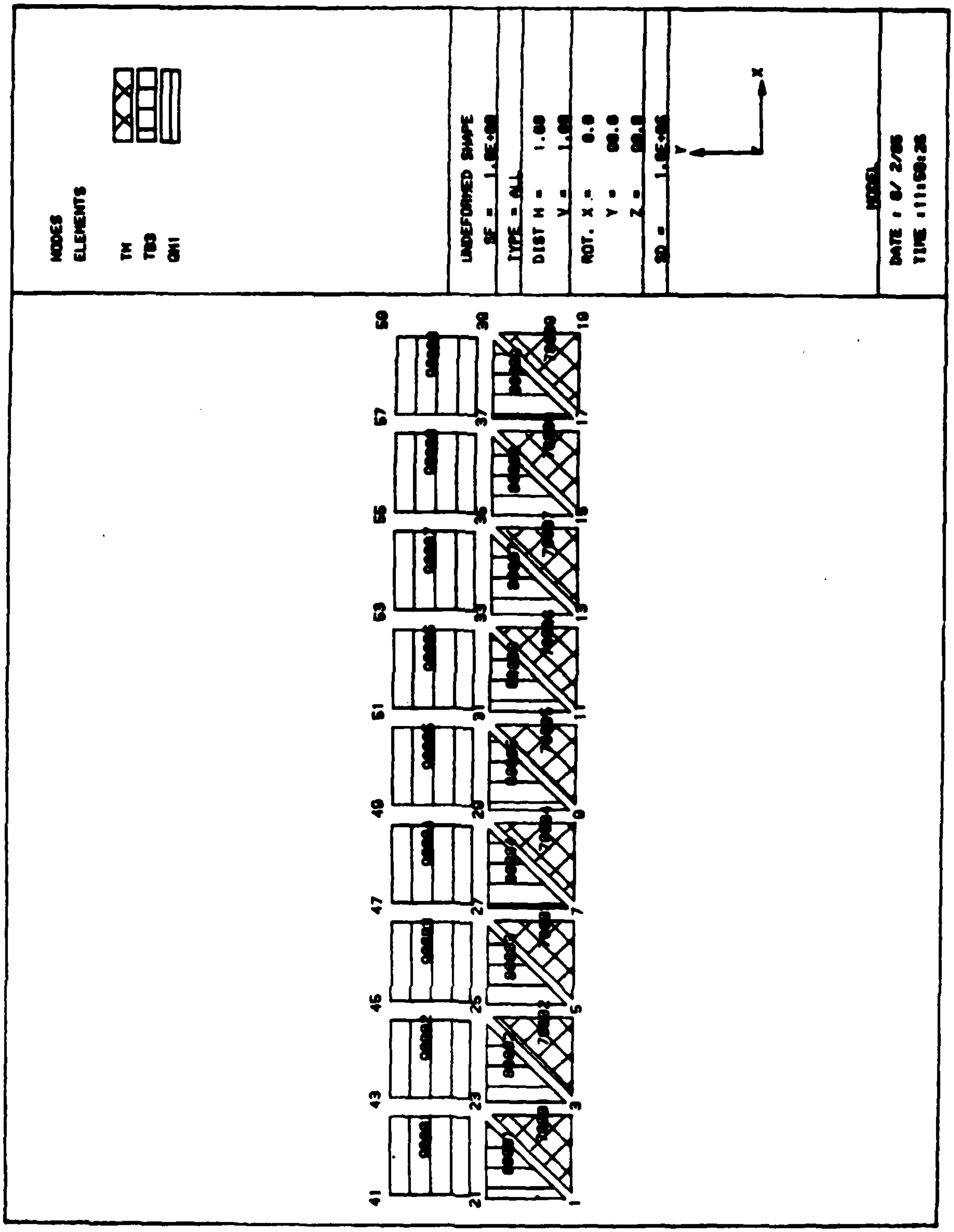
UNCLASSIFIED AFWAL-TR-85-3066-VOL-2 F33615-81-C-3229 F/G 9/2 NL

END

FILMED



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A


NODES
ELEMENTS
TM
TB3
QM1
UNDEFORMED SHAPE
TYPE = ALL
DIST H = 1.00
ROT. X = 0.0
X
Y

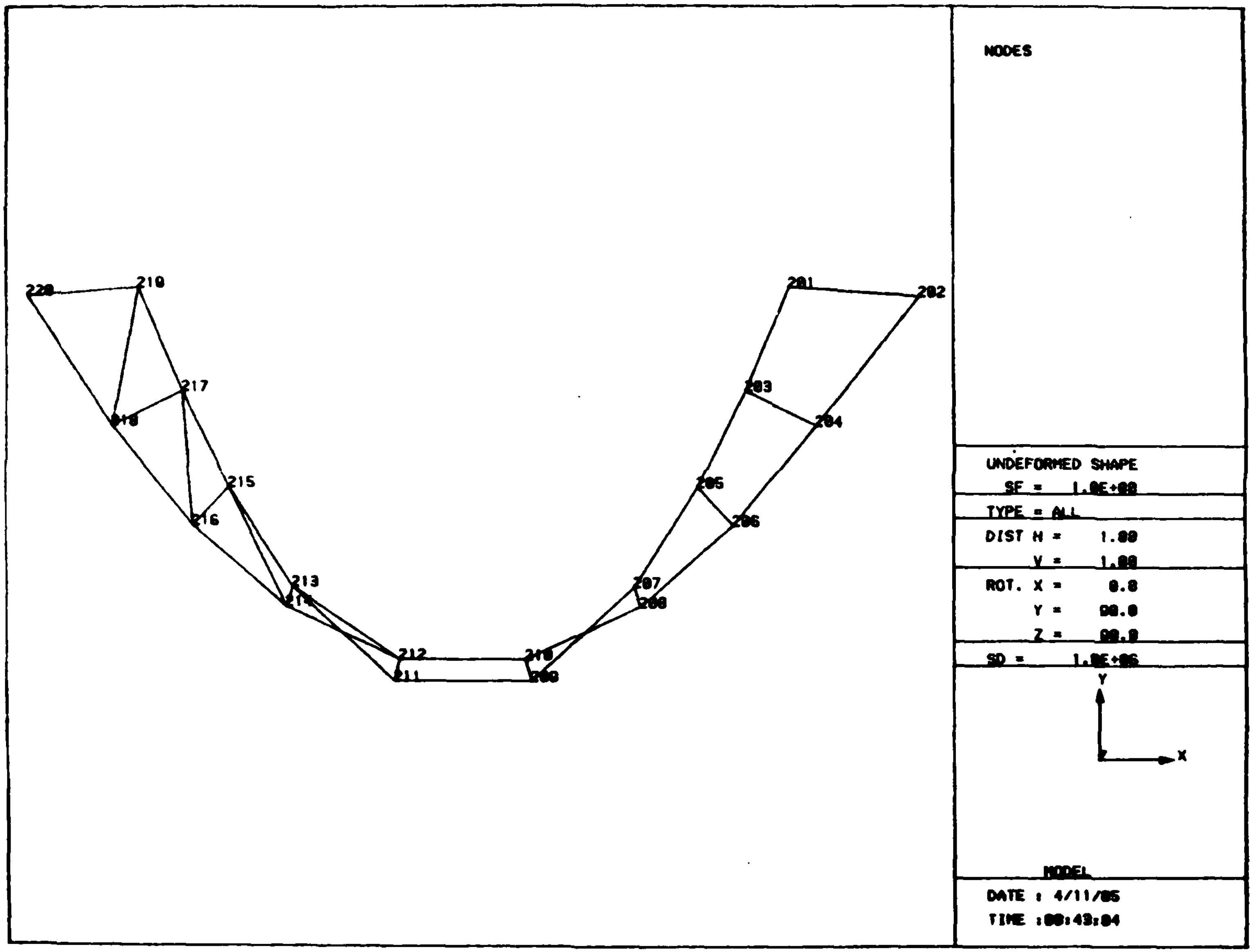
```
 ? CADS

END
 MODULE CADS     ENDED, TIME =00:41:56  DELTA =    1.05
 NATURAL GENERATION STEERING FILE CREATED ON UNIT 3
 CALLED FOR003.DAT.  SHOULD IT BE SAVED (Y/N)?
N
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
N
FORTRAN STOP
$
$
$ RUN CADS
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NATURAL
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
Y
 ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP
GEOMNATU1.DAT
 ? CADS

SET
 ? SET

E1 GROUP 21 22
 ? SET

PLOT E1
```

The PLOT module was ended and CADS was ended for the previous generation case. The Natural Generation Steering file question is an echo of all of the commands entered during use of the NATURAL module. It is normally not saved when a steering file is used as input. The progam was then exited.

Another execution of CADS was started. It attached the previously written geometry data base. The E1 set was built using groups 21 and 22 and the set was sent to the DISPLAY module.

```
 ? DISPLAY
ROTATE Y 90.  Z 90.
 ? DISPLAY
PLOT NODE
```

This set will be used to illustrate the node editor functions under the DISPLAY module. The elements are rotated and then plotted with the node numbers as shown on the next page. Note the cross over of the nodes at the bottom of the plot.



NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 00.0
Z = 00.0
SD = 1.0E+06
Y
Z
X
MODEL
DATE : 4/11/85
TIME : 00:43:04
201
202
203
204
205
206
207
208
209
210
211
212
213
214
215
216
217
218
219
220

```
 ? DISPLAY

EDIT
 ? EDIT

BEGIN NODE
 ? EDITND

LIST NODE 207 TO 214
      207    40.2373     3.4207     0.0000     456
      208    40.4300     2.8434     0.0000     456
      209    37.1063     0.5646     0.0000     456
      210    36.9421     1.2357     0.0000     456
      211    32.8037     0.5646     0.0000     456
      212    33.0570     1.2357     0.0000     456
      213    29.7627     3.4207     0.0000     456
      214    29.5601     2.8434     0.0000     456
 ? EDITND

CHANGE Y NODE 209 211
 ? CHANGE

3.2 3.25
 ? EDITND

END
 MODULE EDITND   ENDED, TIME =08:43:50  DELTA =   97.70
 ? EDIT

END
 MODULE EDIT     ENDED, TIME =08:43:56  DELTA =    6.15
 NOTE: SOME EDIT CHANGES HAVE NOT BEEN SAVED;
       DO YOU STILL WANT TO END (Y/N) ?
N
 ? EDIT

SAVE
 ? EDIT

END
 MODULE EDIT     ENDED, TIME =08:44:16  DELTA =   10.22
 ? DISPLAY

SET
```

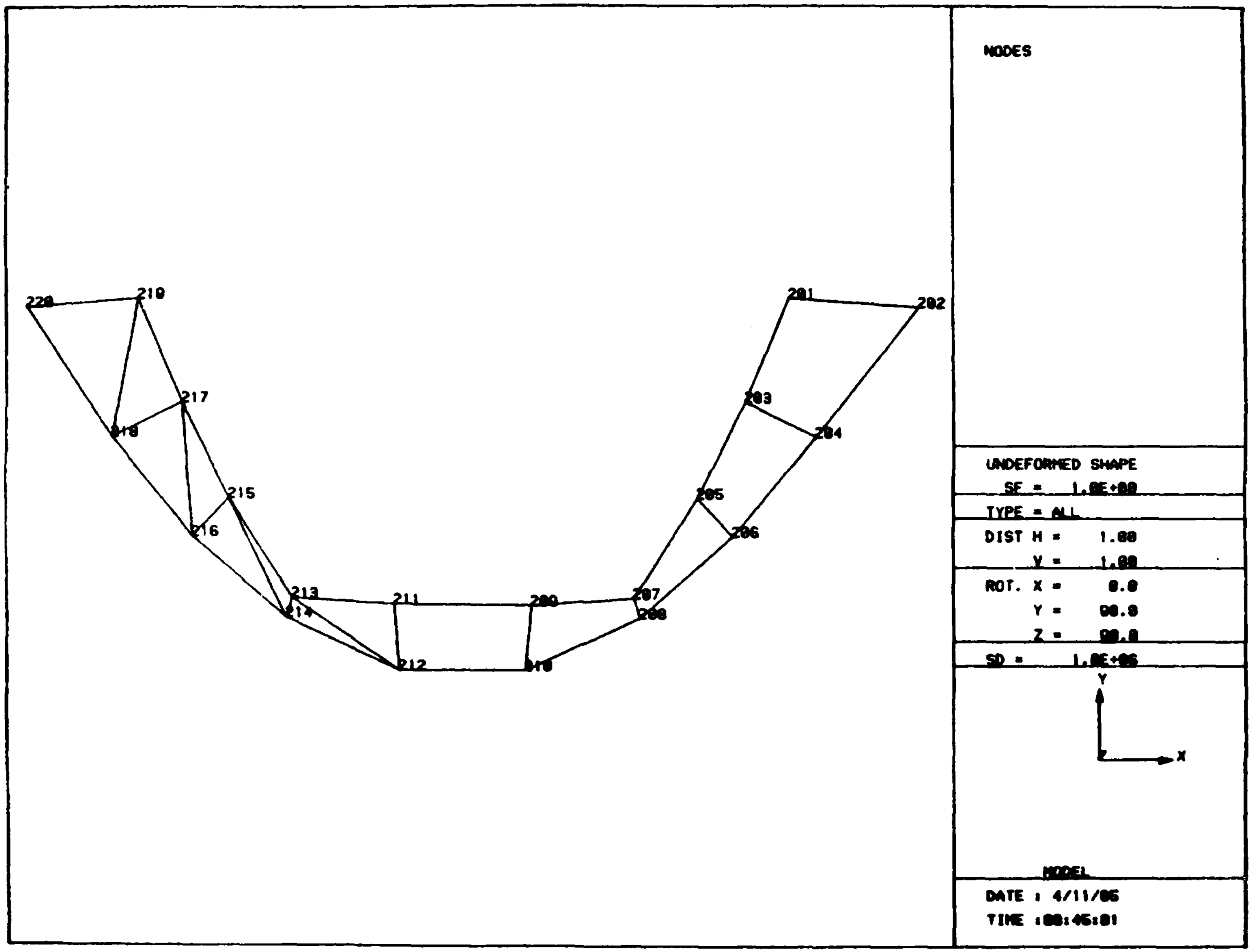
The editor is begun and the BEGIN NODE command starts the node editting functions. The appropriate node data is listed at the terminal using the LIST NODE 207 TO 214 command. The node numbers, coordinates, and single point constraints are listed. The CHANGE Y NODE 209 211 command changes the Y coordinate only for nodes 209 and 211. The new Y terms are entered on the next line as two real numbers using a free format. The END commands end the module. Note the changes were not saved so a warning is issued. The SAVE command then saves the changes and command is returned to the DISPLAY module.

```
? SET
CLEAR ALL
 ? SET
E1 GR 21 22
 ? SET
PLOT E1
```

A SET command was issued from the DISPLAY module to return to the SET module. The current sets are cleared from the active set list and a new set E1 is generated. This set contains groups 21 and 22. It is passed to the DISPLAY module.

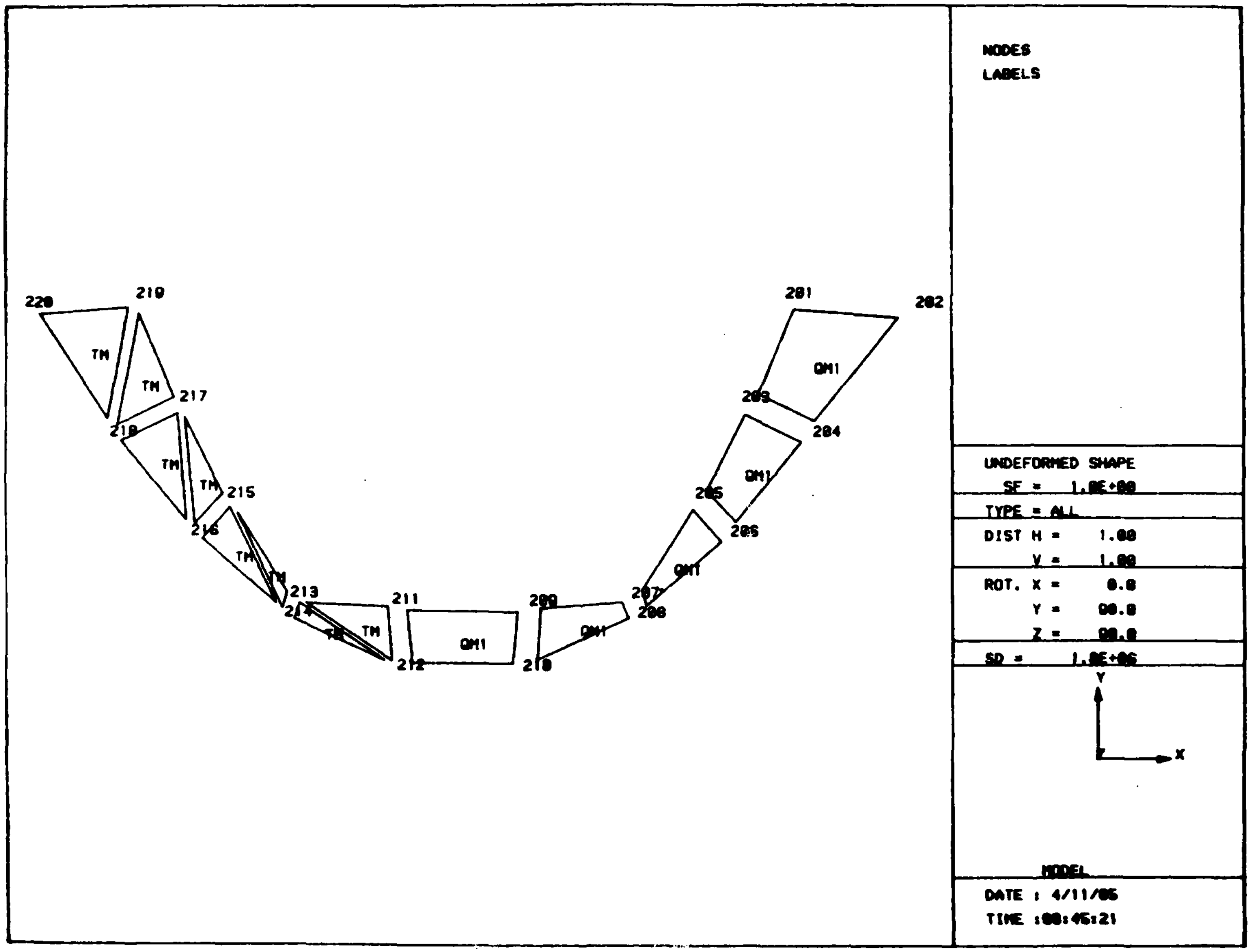
? DISPLAY
PLOT NODE

The elements are plotted with their node numbers. Note the changed position of nodes 209 and 211.


NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 90.0
Z = 90.0
SD = 1.0E+06
Y
X
MODEL
DATE : 4/11/86
201
202
203
204
205
206
207
208
209
210
211
212
213
214
215
216
217
218
219
220

```
? DISPLAY
PL BR ND LA
```

The same set is now displayed with the BREAK, NODE, and LABEL keywords. These keywords were abbreviated using the first two characters of each keyword.


NODES
LABELS
201
202
203
204
205
206
207
208
209
210
211
212
213
214
215
216
217
218
219
220
QM1
TM
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 90.0
Z = 90.0
SD = 1.0E+06
Y
X
Z
MODEL
DATE : 4/11/86
TIME :08:45:21

```
 ? CADS
END
 MODULE CADS     ENDED. TIME =08:46:31  DELTA =    1.06
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
N
FORTRAN STOP
$
```

This model example is ended and control is returned to the VAX at the DEC Control Language (DCL) level.

12.8 NATURAL COMPOSITE WING

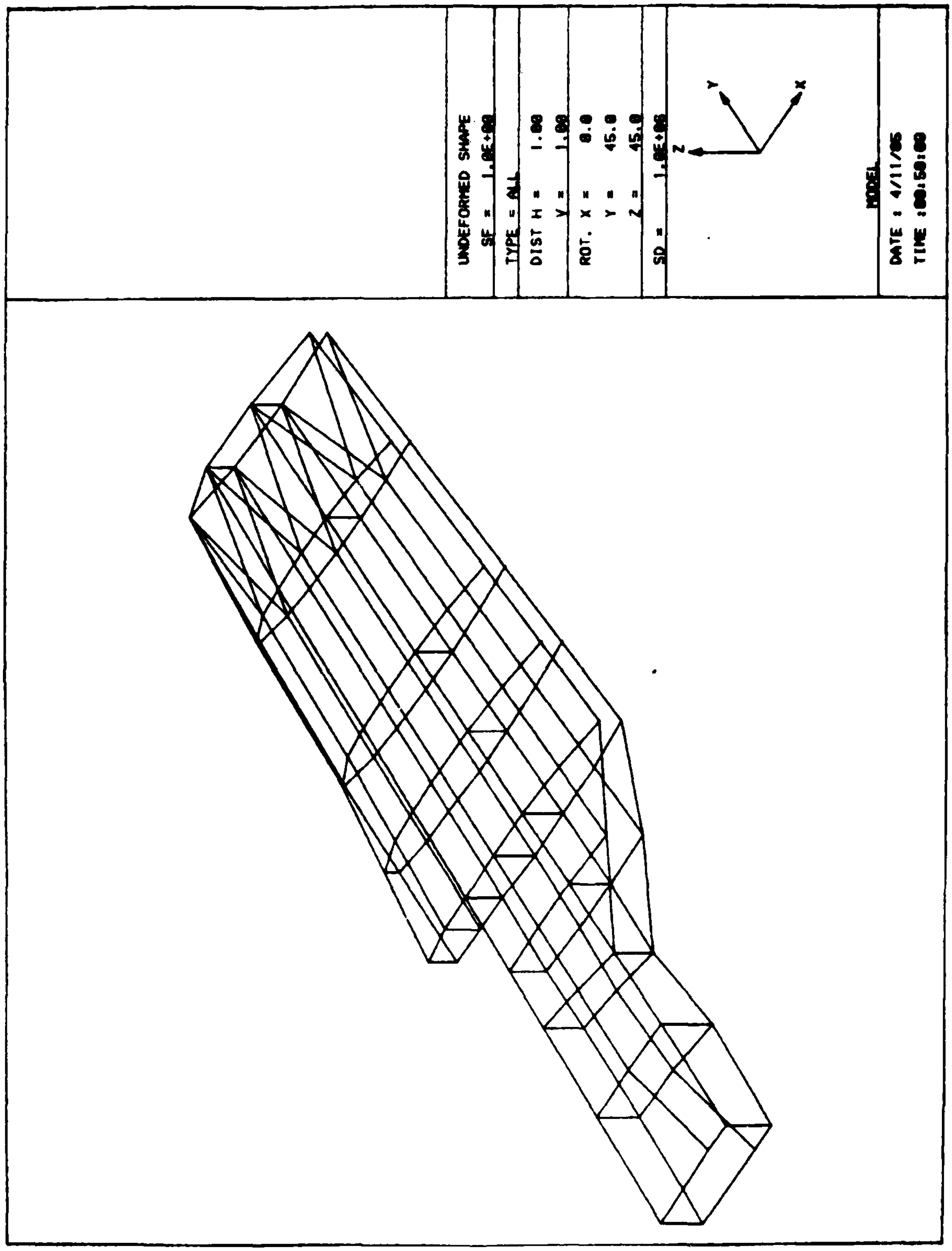
This case is a layered composite wing structure model generated by the NATURAL generation module functions. It is output in the OPTSTAT data requirements. It illustrates the layered composite model generation commands as well as various plotting capabilities.

```
RUN CADS
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NATURAL
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
Y
 ENTER EXISTING GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO SKIP
GEOMNATU2.DAT
 ? CADS

SET
 ? SET

E1 ALL
 ? SET

PLOT E1
```

This is the second example of natural generation. The NATURAL communications mode is used; there is no POST data base; but, there is an existing geometry data base. This data base, GEOMNATU2.DAT, is attached and set E1 is created containing all of the model's elements. It is passed to the DISPLAY module using PLOT E1.

```
 ? DISPLAY
ROTATE Y 45  Z 45
 ? DISPLAY
PLOT
```

The elements are rotated and then plotted using the ROTATE and PLOT commands. This is a simple box structure which will be output in the OPTSTAT data format. The cover elements are composed of layered composite materials.

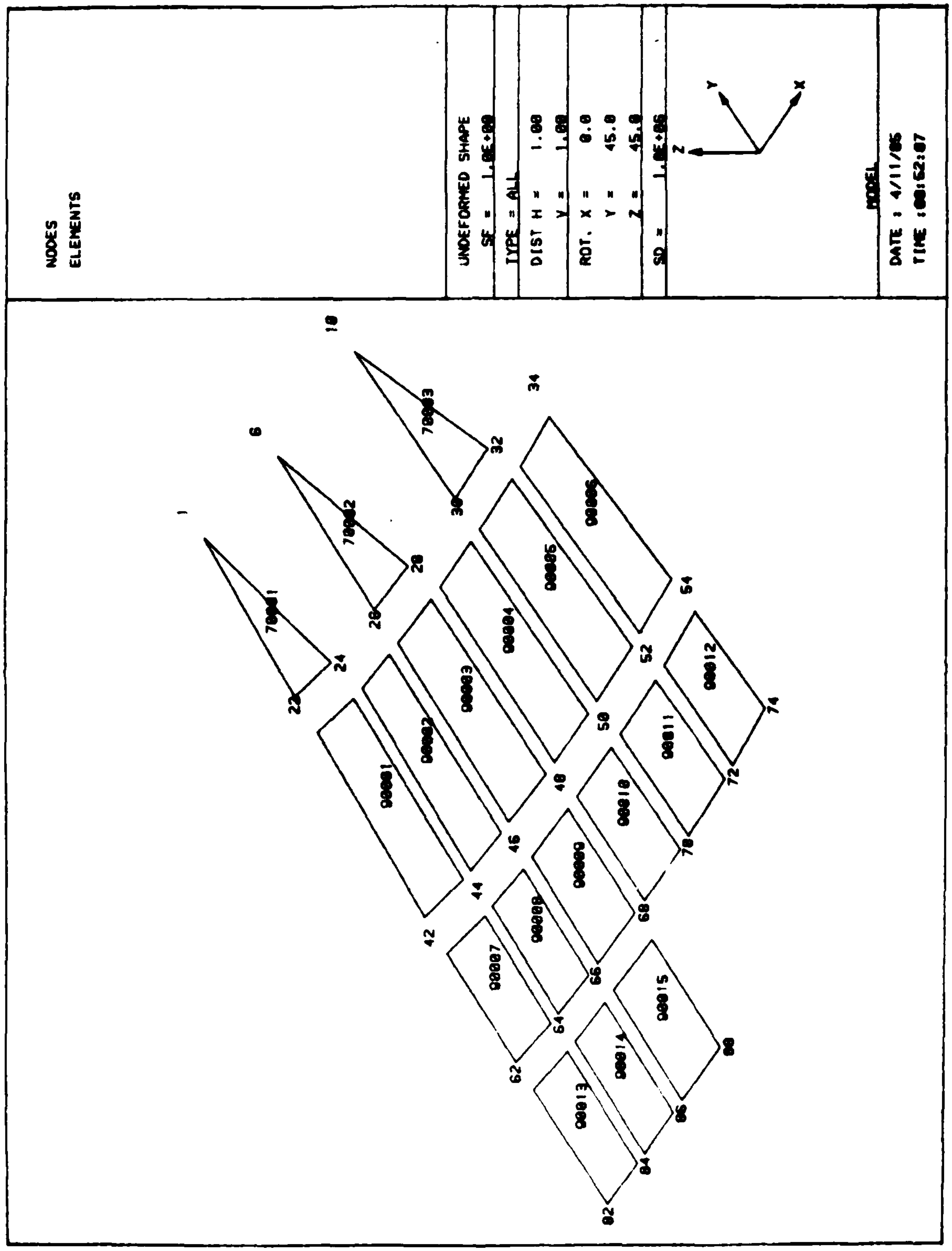

UNDEFORMED SHAPE
TYPE = ALL
DIST H =
V =
ROT. X =
Y =
Z =
SD =
MODEL
DATE :
TIME :

```
? DISPLAY
SET
```

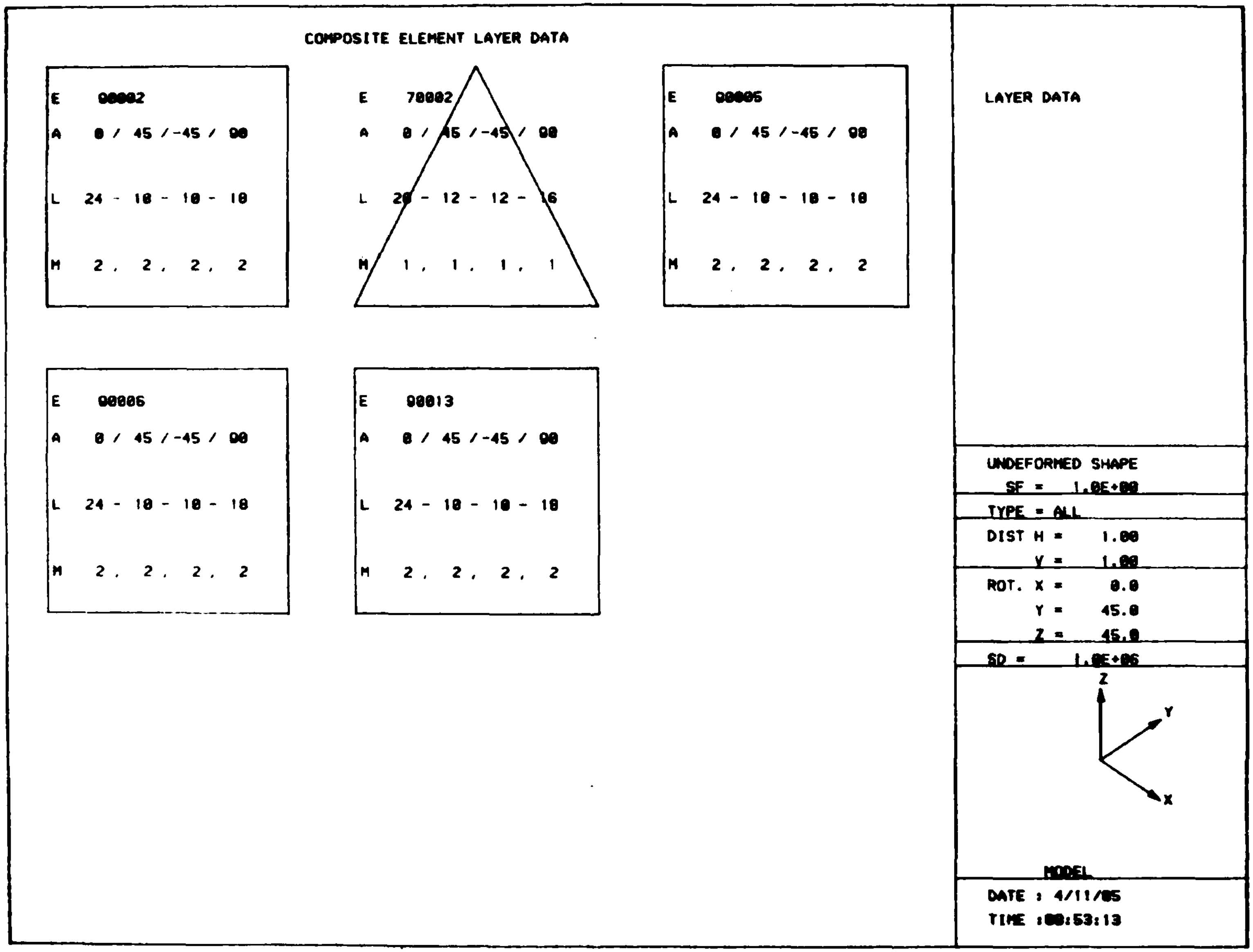
The SET command is issued to return to the SET module. The screen is cleared and the SET prompt comes up.

```
 ? SET

LIST GROUP
   GROUP    1    CSHEAR  = QS4       15  ELEMENTS
   GROUP    2    CSHEAR  = QS4        5  ELEMENTS
   GROUP    3    CTRMEM  = TM         2  ELEMENTS
   GROUP    4    CROD    = CROD      32  ELEMENTS
   GROUP    5    CROD    = CROD      12  ELEMENTS
   GROUP    6    CTRMEM  = TM        11  ELEMENTS
   GROUP    7    CTRMEM  = TM        11  ELEMENTS
   GROUP    8    CQDMEM1 = QM1       27  ELEMENTS
   GROUP    9    CQDMEM1 = QM1       27  ELEMENTS
   GROUP   10    CROD    = CROD      18  ELEMENTS
 ? SET

CLEAR ALL
 ? SET

E1 GROUP 7 EL 1 2 3
 ? SET

E2 GROUP 9 EL 1 TO 15
 ? SET

E1 E1 U E2
 ? SET

PLOT BREAK NODE ELEMENT
```

The model's groups are listed using the LIST GROUP command and the current set list is cleared with the CLEAR command. A set E1 composed of elements 1, 2, and 3 of group 7 is formed as is set E2 composed of elements 1 through 15 in group 9. These two sets are unioned (U) together using the E1 E1 U E2 command. The PLOT command with the BREAK, NODE, and ELEMENT keywords is issued from the SET module. This results in the display on the next page. The PLOT command without a node or element set name will take the last generated element set and display it as if the PLOT command had been issued from the DISPLAY module.


NODES
ELEMENTS
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME : 08:52:07
70001
70002
70003
90001
90002
90003
90004
90005
90006
90007
90008
90009
90010
90011
90012
90013
90014
90015

The detailed composite element material display end of plot processing is now shown. The L character is entered instead of the standard blank and the crosshairs are started. They are then moved to the center of an element for which detailed composite layer data is desired. An integer number (1-9) is entered and the crosshairs are moved to the next element. Up to 9 elements are defined, each with a different number. After the last element is picked an R is entered and the detailed layer data is displayed. This includes the element number, layer angles, number of layers, and material table numbers.

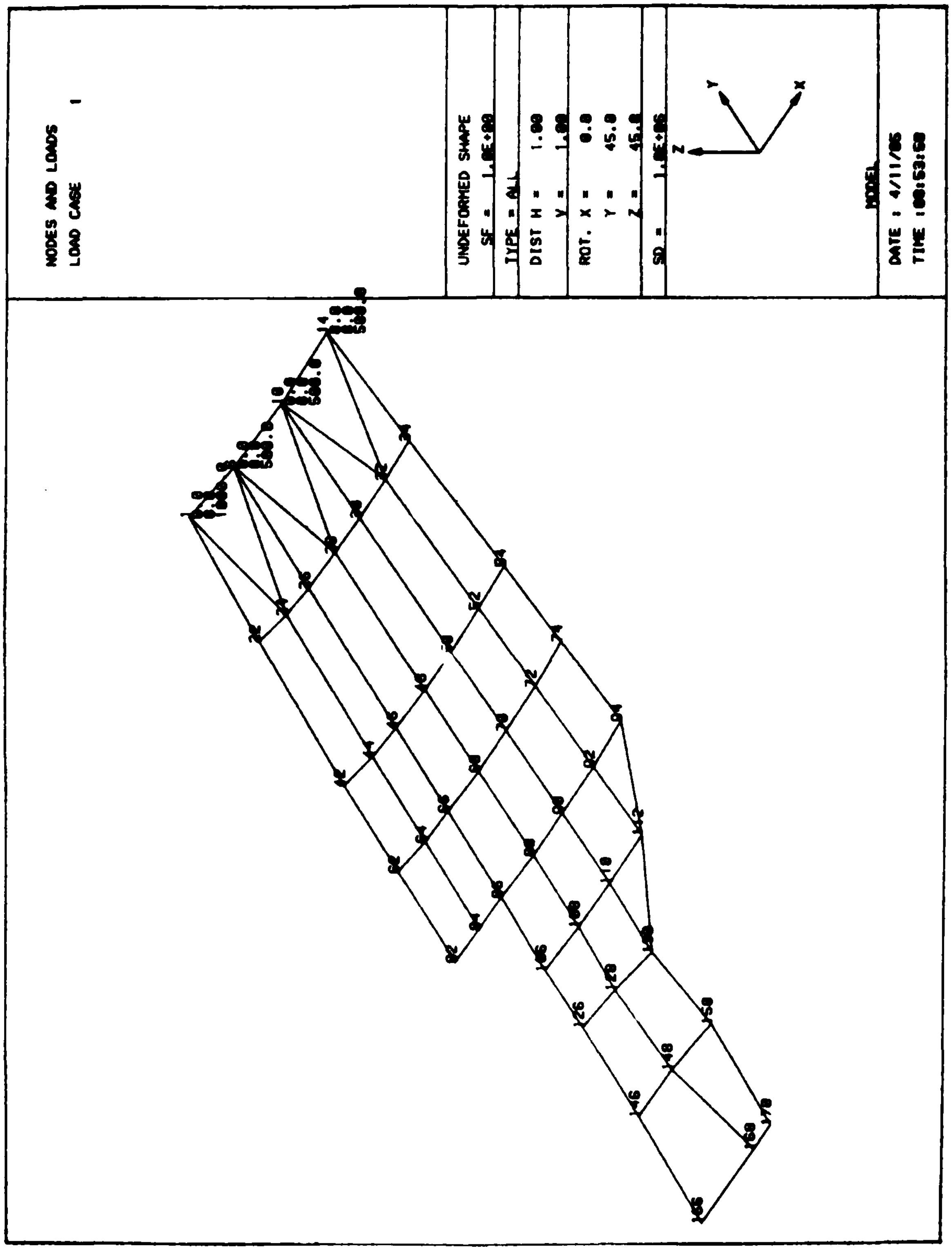

COMPOSITE ELEMENT LAYER DATA
E 00002
A 0 / 45 / -45 / 90
L 24 - 10 - 10 - 10
M 2 , 2 , 2 , 2
E 70002
A 0 / 45 / -45 / 90
L 20 - 12 - 12 - 16
M 1 , 1 , 1 , 1
E 00005
A 0 / 45 / -45 / 90
L 24 - 10 - 10 - 10
M 2 , 2 , 2 , 2
E 00006
A 0 / 45 / -45 / 90
L 24 - 10 - 10 - 10
M 2 , 2 , 2 , 2
E 00013
A 0 / 45 / -45 / 90
L 24 - 10 - 10 - 10
M 2 , 2 , 2 , 2
LAYER DATA
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:53:13

```
? DISPLAY
DEFINE GROUP 7 9
```

A new element set is defined consisting of the model's groups 7 and 9. These groups form the upper cover of this model.

```
? DISPLAY
PLOT NODE LF 1
```

The screen was cleared and the PLOT command was issued. The NODE and LF 1 keywords were then used to display the node numbers and the external load values for load case 1. Note these values on the tip nodes numbered 1, 6, 10, and 14.



NODES AND LOADS
LOAD CASE 1
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
MODEL

12.9 NATURAL ISOTROPIC WING

This case is an isotropic wing structure model generated by the NATURAL module functions. It is output in the ANALYZE data format and illustrates various grid point, load, element, and material definition commands as well as various plotting capabilities.

```
 ? CADS

OUTPUT
 ? OUTPUT

BEGIN OPTSTAT
 ENTER OPTSTAT OUTPUT FILE NAME NOW OR END TO STOP
OPTBULK.DAT
 FILE OPTBULK.DAT
 ALREADY EXISTS SHOULD IT BE REUSED (Y/N) ?
Y
 ? OUTPUT

END
 MODULE OUTPUT   ENDED. TIME =08:54:37  DELTA =   35.16
 ? CADS

END
 MODULE CADS     ENDED. TIME =08:54:40  DELTA =    3.56
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
Y
 ENTER THE TERMINAL BEING USED.
 VALID TYPES : ALPHA , 4014 , CALC
4014
 ENTER BAUD RATE FOR TERMINAL AS 300, 1200, ..., 19200
 (THIS IS A HARDWARE REQUIREMENT - DEFAULT IS 9600)
0
 ENTER THE PROGRAM COMMUNICATION TYPE:
 (RESPOND EITHER : NASTRAN, ANALYZE, NATURAL, OR OPTSTAT
 ? START

NATURAL
 HAVE YOU ATTACHED A POST-PROCESSING FILE (Y/N)
N
 DO YOU HAVE AN EXISTING DATABASE  (Y/N)
N
 ENTER THE TITLE TO MODEL HEADER
TEST NATURAL ANALYZE INPUT
 ENTER NEW GEOMETRY DATA BASE FILE NAME FOR CADS OR END TO STOP
GEOMNATU3.DAT
 ? CADS

READ
 ? READ

BEGIN NATURAL INPUT 20
 ENTER NATURAL INPUT FILE NAME NOW OR END TO RETURN
NATUIN3.DAT
```

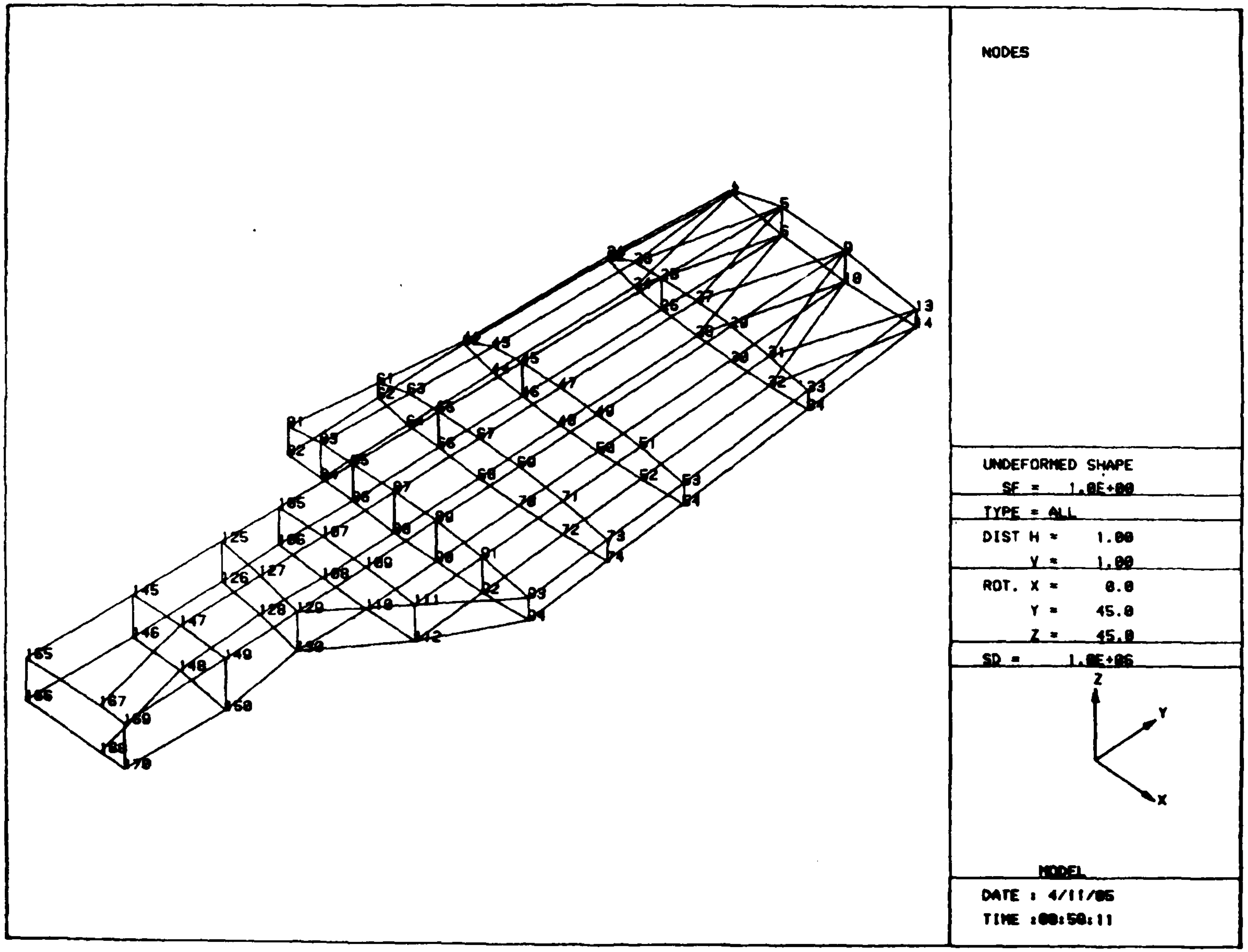
Control was returned to CADS so that the OUTPUT module could be started by the OUTPUT command. The geometry data on the data base was translated to the OPTSTAT format and stored in the OPTBULK.DAT file using the BEGIN OPTSTAT command. CADS was then ended and the third generation example was started. This was to create a new geometry data base (GEOMNATU3.DAT). The NATUIN3.DAT file was read in under the BEGIN NATURAL IN 20 command.

```
 MODULE DIRECT   ENDED, TIME =08:58:14  DELTA =  37.030
 MODULE FREEDOM  ENDED, TIME =08:58:14  DELTA =  38.131
 MODULE LOAD     ENDED, TIME =08:58:15  DELTA =  38.420
 MODULE NODES    ENDED, TIME =08:58:15  DELTA =   38.40
 MODULE ELEMENT  ENDED, TIME =08:58:16  DELTA =   1.541
 MODULE DIRECT   ENDED, TIME =08:58:19  DELTA =   4.270
 MODULE PROPERTY ENDED, TIME =08:58:19  DELTA =   4.420
 MODULE NATURAL  ENDED, TIME =08:58:19  DELTA =    4.40
      9  ACCEPTABLE GROUPS PROCESSED

  GROUP    1     CSHEAR  = QS4          16  ELEMENTS
  GROUP    2     CSHEAR  = QS4           9  ELEMENTS
  GROUP    4     CROD    = CROD         36  ELEMENTS
  GROUP    5     CROD    = CROD         18  ELEMENTS
  GROUP    6     CTRMEM  = TM           11  ELEMENTS
  GROUP    7     CTRMEM  = TM           11  ELEMENTS
  GROUP    8     CQDMEM1 = QM1          27  ELEMENTS
  GROUP    9     CQDMEM1 = QM1          27  ELEMENTS
  GROUP   10     CROD    = CROD         24  ELEMENTS
 ? READ

END
 MODULE READ     ENDED. TIME =08:58:24  DELTA =    4.98
 ? CADS

SET
 ? SET

E1 ALL
 ? SET

PLOT E1
```

The model's groups are listed; the READ module is ended; and an element set is defined. This set contains all of the elements and is passed to the DISPLAY module using the PLOT E1 command.

```
? DISPLAY
ROTATE  Y 45  Z 45
 ? DISPLAY
PLOT NODE
```

The elements are rotated and plotted with their node numbers using the ROTATE and PLOT commands. This model is very similar to the previous example with the exception that it is composed of isotropic materials and will be output in the ANALYZE data format.

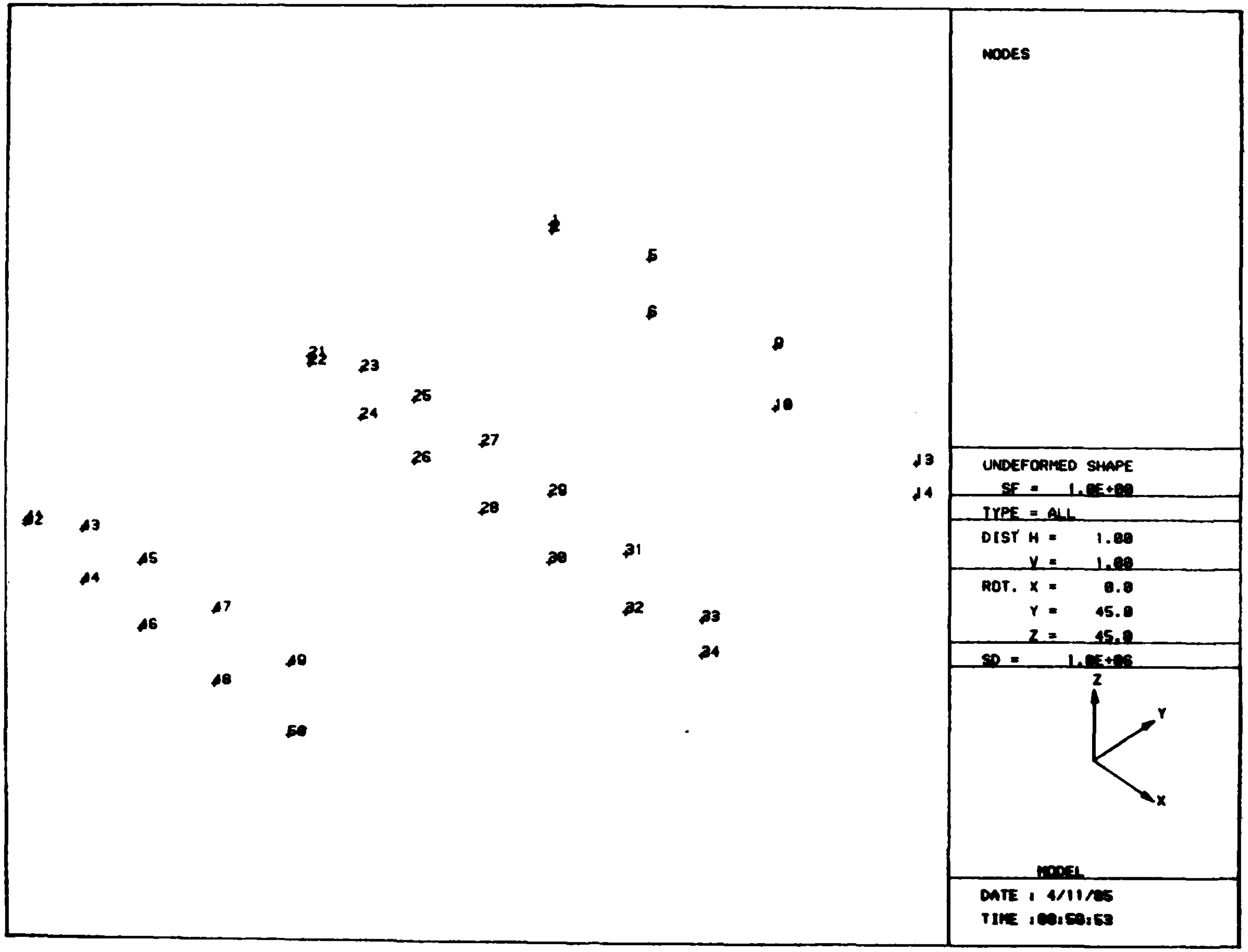

NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :08:50:11

```
? SET
N1 1 TO 50
? SET
PLOT N1
```

Control was returned to SET with the RETURN command. A node set, named N1, is specified with the nodes numbered from 1 through 50 in the set. The N1 set is passed to the DISPLAY module using the PLOT N1 command. Note node set names start with N; element set names start with E.

```
? DISPLAY
PLOT NODE
```

The nodes are now plotted as + marks along with the node numbers. The PLOT command with the NODE keyword performs this function.

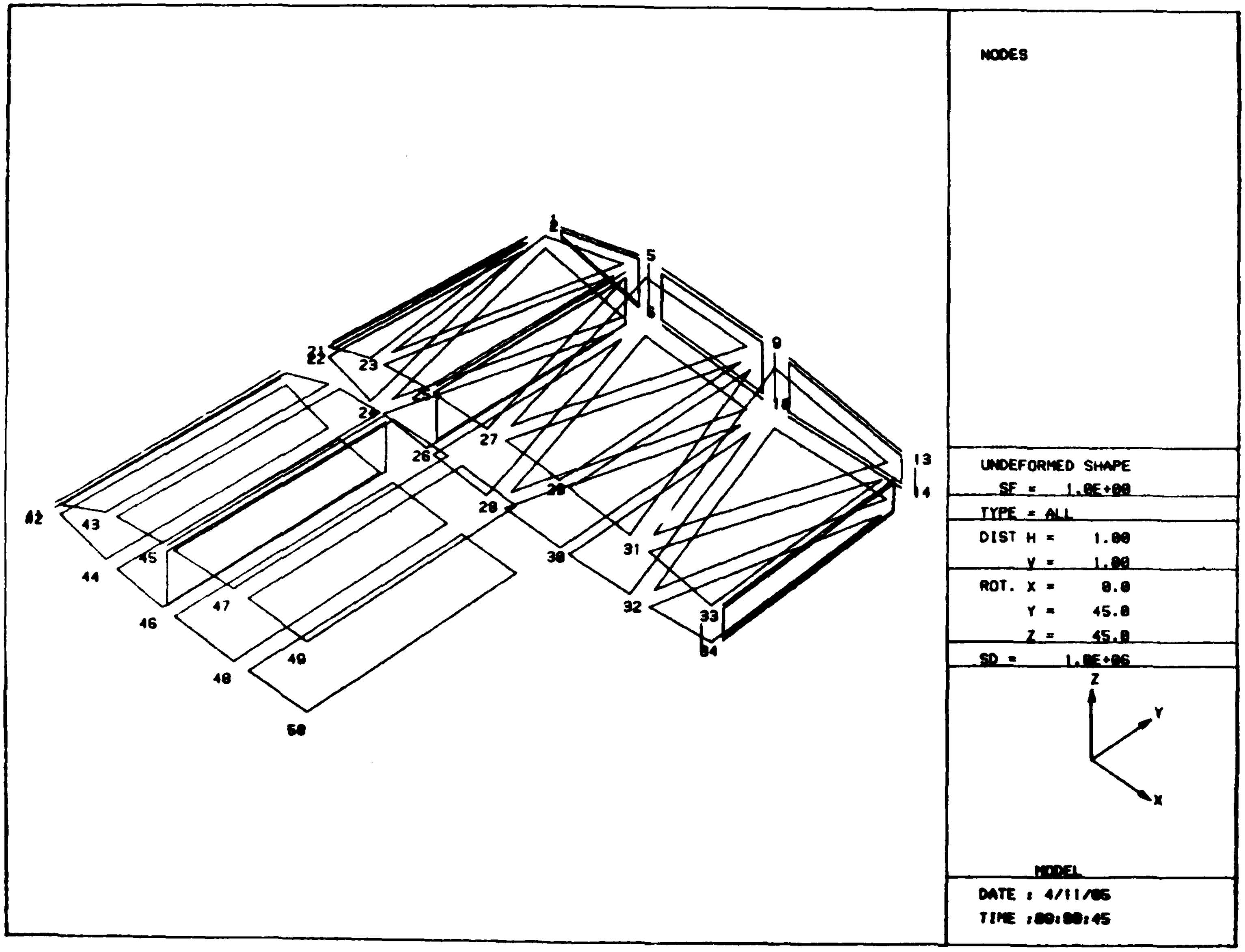

NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/85
TIME :00:58:53

```
? SET
N1 1 TO 60
? SET
E1 N1
? SET
PLOT E1
```

Control is returned to SET and the node set N1 is generated. An element set E1 is then generated from the node set N1. Set E1 will contain all of the elements which have all of their nodes within the node set N1. For example, if an element is composed of nodes 30, 40, 50, and 60 it would not be in the set E1 since node 60 is not in the node set N1. Set E1 is then passed to the DISPLAY module.

```
? DISPLAY
PLOT BREAK NODE
```

The element set is now plotted using the BREAK and NODE keywords on the PLOT command.


NODES
UNDEFORMED SHAPE
SF = 1.0E+00
TYPE = ALL
DIST H = 1.00
V = 1.00
ROT. X = 0.0
Y = 45.0
Z = 45.0
SD = 1.0E+06
Z
Y
X
MODEL
DATE : 4/11/86
TIME :00:00:45

```
 ? CADS

OUTPUT
 ? OUTPUT

BEGIN ANALYZE
 ENTER ANALYZE OUTPUT FILE NAME NOW OR END TO STOP
ANALBULK.DAT
 ? OUTPUT

END
 MODULE OUTPUT   ENDED. TIME =13:57:08  DELTA =   60.11
 ? CADS

END
 MODULE CADS     ENDED, TIME =13:57:15  DELTA =    6.45
 NATURAL GENERATION STEERING FILE CREATED ON UNIT 3
 CALLED FOR003.DAT.  SHOULD IT BE SAVED (Y/N)?
N
 DO YOU WISH TO PROCESS ANOTHER MODEL (Y/N) ?
N
FORTRAN STOP
$
```

Control was returned to CADS and the OUTPUT module was started. The BEGIN ANALYZE command will then output the geometry data in the ANALYZE data format. The data will be written to the file: ANALBULK.DAT. The OUTPUT module was then ended as was CADS. Again the natural steering file can be saved or another model can be started. In this case control was returned to the host DEC VAX machine.

12.10 CADSPP TEST

These test cases are a sample of the loading procedure for taking analysis program output data and loading it into a post data base.

```
RUN CADSPP
 ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP
OPTOUT.DAT
 ENTER TYPE OF ANALYSIS DATA BEING STORED; NASTRAN, ANALYZE, OPTSTAT
    OR ENTER END TO STOP
OPTSTAT
 IS A NEW POST DATA BASE TO BE GENERATED (Y/N) ?
Y
  ENTER FILE NAME FOR POST DATA BASE
POSTOPT.DAT
SPECIFY DATA BLOCK TYPES TO BE STORED  DEFAULT IS ALL
STRESS  DISPLACE
 OPTSTAT DATA READ AND STORED
 ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP
END
FORTRAN STOP
$
```

This is a sample execution of the CADSPP program. The executable is named CADSPP. The program prompts for the analysis output data file name, in this case OPTOUT.DAT. Next it asks for the type of data being processed which was OPTSTAT and if a new POST data base is being generated. In this case the new data base was named POSTOPT.DAT. The STRESS and DISPLACEMENT data was to be read from the OPTOUT.DAT file and converted for storage on the POSTOPT.DAT data base. This was completed and CADSPP prompted for another analysis file name or end to stop. An END was entered and control returns to the host VAX.

```
RUN CADSPP
 ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP
NASTOUT.DAT
 ENTER TYPE OF ANALYSIS DATA BEING STORED; NASTRAN, ANALYZE, OPTSTAT
    OR ENTER END TO STOP
NASTRAN
 IS A NEW POST DATA BASE TO BE GENERATED (Y/N) ?
Y
  ENTER FILE NAME FOR POST DATA BASE
POSTNAST3.DAT
ENTER INPUT TYPE STATIC OR DYNAMIC WITH TIME INC ( STATIC/DYNAMIC
STATIC
SPECIFY DATA BLOCK TYPES TO BE STORED  DEFAULT IS ALL
STRESS FORCE  DISPLACE
 ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP
END
FORTRAN STOP
$
```

This is an execution of CADSPP. The program prompts for the analysis output data file name, in this case NASTOUT.DAT. Next it asks for the data type being processed which is NASTRAN and if a new POST data base is being generated. In this case the new data base is named POSTNAST3.DAT. CADSPP then prompts for STATIC or DYNAMIC to determine the conversion procedure for reading the input analysis data. The STRESS, FORCE, and DISPLACEMENT data is read from the NASTOUT.DAT file and stored on the POSTNAST3.DAT data base. Finally CADSPP prompted for another analysis file name before it was ended and control returned to the VAX.

```
RUN CADSPP
 ENTER FILE NAME FOR ANALYSIS PROGRAM OUTPUT DATA OR END TO STOP
NASTTIME.DAT
 ENTER TYPE OF ANALYSIS DATA BEING STORED: NASTRAN, ANALYZE, OPTSTAT
    OR ENTER END TO STOP
NASTRAN
 IS A NEW POST DATA BASE TO BE GENERATED (Y/N) ?
Y
  ENTER FILE NAME FOR POST DATA BASE
POSTTIME.DAT
ENTER INPUT TYPE STATIC OR DYNAMIC WITH TIME INC ( STATIC/DYNAMIC )
DYNAMIC
SPECIFY DATA BLOCK TYPES TO BE STORED  DEFAULT IS ALL
DISPLACE  STRESS
FORTRAN STOP
$
```

This is an execution of CADSPP. The program prompts for the analysis output data file name, in this case NASTTIME.DAT. Next it asks for the data type being processed which is NASTRAN and if a new POST data base is being generated. In this case the new data base is named POSTTIME.DAT. CADSPP then prompts for STATIC or DYNAMIC to determine the conversion procedure for reading the input analysis data. The dynamic DISPLACEMENT and STRESS data is then read from the NASTTIME.DAT file and stored on the POSTTIME.DAT data base. Finally CADSPP prompted for another analysis file name before it was ended and control returned to the VAX.

REFERENCES

1. DI-3000 User's Guide, Precision Visuals, Incorporated, Boulder, Colorado, April 1981.

2. DI-3000 Installation Notes, Precision Visuals, Incorporated, Boulder, Colorado, October 1981.

3. NASA Reference 1085, COSMIC No. ARC-11446, A General Solution to the Hidden-Line Problem, Hedgley, David R., Jr., Ames Research Center, Dryden Flight Research Facility, Edwards, California, 1982.

4. ANSI X3.9-1978, American National Standard Programming Language FORTRAN, April 1978.

5. VAX-11 FORTRAN User's Guide, Digital Equipment Corporation, Maynard, Massachusetts, April 1980.

6. VS FORTRAN Application Programming: Language Reference, International Business Machines Corporation, September 1982.

7. Venkayya, V. B., and Tischler, V. A., "ANALYZE - Analysis of Aerospace Structures with Membrane Elements," AFFDL-TR-78-170.

8. Venkayya, V. B., and Tischler, V. A., "OPTSTAT - A Computer Program for the Optimal Design of Structures Subjected to Static Loads," TM-FBR-79-67.

☆ [illegible] 067 067 40639

END

FILMED

2-86

DTIC